IPC
ASSOCIATION CONNECTING ELECTRONICS INDUSTRIES®

IPC-HDBK-005

Guide to Solder Paste Assessment

A standard developed by IPC

3000 Lakeside Drive, Suite 309 S, Bannockburn, IL 60015-1249
Tel. 847-615-7100 Fax 847-615-7105
www.ipc.org

The Principles of Standardization

In May 1995 the IPC's Technical Activities Executive Committee (TAEC) adopted Principles of Standardization as a guiding principle of IPC's standardization efforts.

Standards Should:
- Show relationship to Design for Manufacturability (DFM) and Design for the Environment (DFE)
- Minimize time to market
- Contain simple (simplified) language
- Just include spec information
- Focus on end product performance
- Include a feedback system on use and problems for future improvement

Standards Should Not:
- Inhibit innovation
- Increase time-to-market
- Keep people out
- Increase cycle time
- Tell you how to make something
- Contain anything that cannot be defended with data

Notice

IPC Standards and Publications are designed to serve the public interest through eliminating misunderstandings between manufacturers and purchasers, facilitating interchangeability and improvement of products, and assisting the purchaser in selecting and obtaining with minimum delay the proper product for his particular need. Existence of such Standards and Publications shall not in any respect preclude any member or nonmember of IPC from manufacturing or selling products not conforming to such Standards and Publication, nor shall the existence of such Standards and Publications preclude their voluntary use by those other than IPC members, whether the standard is to be used either domestically or internationally.

Recommended Standards and Publications are adopted by IPC without regard to whether their adoption may involve patents on articles, materials, or processes. By such action, IPC does not assume any liability to any patent owner, nor do they assume any obligation whatever to parties adopting the Recommended Standard or Publication. Users are also wholly responsible for protecting themselves against all claims of liabilities for patent infringement.

IPC Position Statement on Specification Revision Change

It is the position of IPC's Technical Activities Executive Committee that the use and implementation of IPC publications is voluntary and is part of a relationship entered into by customer and supplier. When an IPC publication is updated and a new revision is published, it is the opinion of the TAEC that the use of the new revision as part of an existing relationship is not automatic unless required by the contract. The TAEC recommends the use of the latest revision. Adopted October 6, 1998

Why is there a charge for this document?

Your purchase of this document contributes to the ongoing development of new and updated industry standards and publications. Standards allow manufacturers, customers, and suppliers to understand one another better. Standards allow manufacturers greater efficiencies when they can set up their processes to meet industry standards, allowing them to offer their customers lower costs.

IPC spends hundreds of thousands of dollars annually to support IPC's volunteers in the standards and publications development process. There are many rounds of drafts sent out for review and the committees spend hundreds of hours in review and development. IPC's staff attends and participates in committee activities, typesets and circulates document drafts, and follows all necessary procedures to qualify for ANSI approval.

IPC's membership dues have been kept low to allow as many companies as possible to participate. Therefore, the standards and publications revenue is necessary to complement dues revenue. The price schedule offers a 50% discount to IPC members. If your company buys IPC standards and publications, why not take advantage of this and the many other benefits of IPC membership as well? For more information on membership in IPC, please visit www.ipc.org or call 847/597-2872.

Thank you for your continued support.

Acknowledgment

Any document involving a complex technology draws material from a vast number of sources. While the principal members of the Solder Paste Task Group (5-24b) of the Assembly and Joining Processes Committee (5-20) are shown below, it is not possible to include all of those who assisted in the evolution of this standard. To each of them, the members of the IPC extend their gratitude.

Assembly and Joining Processes Committee

Chair
Leo P. Lambert
EPTAC Corporation

Solder Paste Task Group

Co-Chairs
Paul Niemczura
Heraeus, Inc.

Brian J. Toleno, Ph.D
Henkel Corporation

Technical Liaisons of the IPC Board of Directors

Peter Bigelow
IMI Inc.

Sammy Yi
Flextronics International

Solder Paste Task Group

David C. Adams, Rockwell Collins
Patricia J. Amick, Boeing Aircraft & Missiles
Scott J. Anson, PE, Rochester Institute of Technology
Victor P. Balasbas, Jr., Qualitek International Inc.
Victor J. Barba, Endicott Interconnect Technologies Inc
Allan Beikmohamadi, E. I. du Pont de Nemours and Co.
Ross Berntson, Indium Corporation of America
Richard W. Boerdner, EJE Research
Rex L. Breunsbach, Electronic Controls Design Inc. (ECD)
Thomas A. Carroll, Boeing Space Systems
Srinivas Chada, Ph.D, Jabil Circuit, Inc.
Beverley Christian, Ph.D., Research In Motion Limited
Jerry Cupples, VLSIP Technologies
Charles Dal Currier, Ambitech Inc.
Glenn Dody, Dody Consulting
Karen Walters Ebner, Raytheon Company
Gary J. Ewell, The Aerospace Corporation
Joe R. Felty, Raytheon Company
Tom Fujikawa, Malcomtech International
Alan Gickler, Johnson Manufacturing Co.
Bruce W. Gray, EMC Corporation
Hue T. Green, Lockheed Martin Space Systems Company
Leslie A. Guth, Ph.D., Lucent Technologies, Inc.
Matt Hamand, Rockwell Collins
Charles Han, Lace Technologies
David D. Hillman, Rockwell Collins
Helen Holder, Hewlett-Packard Company
Hana Hsu, Mitac International Corporation
Frank Hules, Advanced Motion Controls
Christopher Hunt, Ph.D., National Physical Laboratory
Jennie S. Hwang, Asahi Technologies America, Inc.
Les Hymes, The Complete Connection
Tim Jensen, Indium Corporation of America
Alden C. Johnson, Speedline Technologies, Inc.
Kathryn L. Johnson, Hexacon Electric Company
Prakash Kapadia, Celestica International Inc.
Donald Karp, Trace Laboratories - Central
William G. Kenyon, Global Centre Consulting
Arshad Khan, Celestica International Inc.
Vincent B. Kinol, Umicore America Inc.
Connie M. Korth, Reptron Manufacturing Services/Hibbing
Richard E. Kraszewski, Kimball Electronics
Vijay Kumar, Lockheed Martin Missile & Fire Control
Mark A. Kwoka, Intersil Corporation
Tim Lawrence, Multicore Solders Ltd.
Ning-Cheng Lee, Ph.D., Indium Corporation of America
Kuan-Shaur Lei, Hewlett-Packard Company
Paul Lotosky, Alpha, Cookson Electronics
James F. Maguire, Intel Corporation
Douglas B McKernan, Amtech, Inc.
Pauline McNaught, EFD Inc., Solder Paste Group
Jay Messner, The Boeing Company
Renee J. Michalkiewicz, Trace Laboratories - East
Anton Zoran Miric, W. C. Heraeus GmbH & Co. KG
Benny Nilsson, Ericsson AB
Fraser Nimmo, Qualitek International Inc.

Debora L. Obitz, Trace Laboratories - East

George Oxx, Solectron Technology Inc.

Deepak K. Pai, C.I.D.+, General Dynamics-Advanced Information

Stephen Paradis, Stecoah Systems

John M. Radman, Trace Laboratories - East Denver

John H. Rohlfing, Delphi Electronics and Safety

Paul J. Roselle, Lockheed Martin Maritime Systems

David Sbiroli, Indium Corporation of America

David F. Scheiner, Kester

Karl F. Seelig, AIM, Inc.

Quan Sheng, Umicore America Inc.

James A. Slattery, Indium Corporation of America

John E. Sohn Ph.D., Consultant

John R. Sovinsky, Indium Corporation of America

Jorgen Svensson, Sony Ericsson Mobile Communications AB

Keith Sweatman, Consultant

Karen A. Tellefsen, Ph.D., Alpha, Cookson Electronics

Dung Q. Tiet, Lockheed Martin Space Systems Company

Ineke Van Tiggelen, Cobar Europe BV

Kristen K. Troxel, Hewlett-Packard Company

Laura J. Turbini, Ph.D., University of Toronto

Malcolm Warwick, Multicore Solders Ltd.

Tony West, Plexus Electronic Assembly Corporation

Eli Westerlaken, Cobar Europe BV

Dewey Whittaker, Honeywell Inc.

Todd Woods, Plexus NPI Plus

Ailan Zhu, Huawei Technologies Co., Ltd.

Table of Contents

Figures

Tables

Guide to Solder Paste Assessment

1 SCOPE

This handbook is a companion to the solder paste standard J-STD-005 and should be considered to be a guide to help assess the applicability of a solder paste for its use in surface mount technology (SMT) processes. This document also suggests some test methods that can help with designing and testing solder pastes. It is intended for use by both vendors and users of solder paste.

1.1 Purpose Solder pastes are unique materials, whose performance in a surface mount process depends on a variety of variables, many of them interacting. J-STD-005 provides test methods for classification of solder paste based on the use of a variety of testing techniques. However, these solder paste classifications do not have a direct correlation to identify the type and characteristics of a specific solder paste that is needed in any given SMT assembly process.

This document has been written as a guide to assess the applicability of a solder paste for a specific process, given the tremendous number of permutations of different materials, atmospheres and process variables currently available.

Where appropriate, references are given to papers and documents with further information. Due to the sheer number of possible interacting factors, specific solder paste selection criteria cannot be given. The solder paste selected and the assembly process used will need to form solder connections that meet the requirements of industry standards such as J-STD-001 and/or IPC-A-610.

1.2 Checklist Solder paste vendors often provide practical information about their products. When this information is available much time and money can be saved in qualifying the product's capabilities. The following listings are examples of information that is usually of value to process engineers and designers. Items marked with an asterisk (*) are usually of greatest value.

*Is the paste recommended for use in clean or no-clean processes and is there a recommended cleaning process?
*What is the ideal or recommended reflow profile?
*What is the flux classification per J-STD-004 or equivalent flux standard?

- *What are the results of copper mirror testing?
- *What is the halides content?
- *What percentage of Flux Solids (Non-Volatile Content)?
- *Corrosion
- *Surface Insulation Resistance (SIR)

*What is the characterization of solder paste per J-STD-005?

- *Powder Size
- *Powder Shape
- *Metal Percent
- *Viscosity
- *Slump
- *Solder Ball
- *Tack
- *Wetting

*What is the solder powder alloy designation per J-STD-006?

- Acid Value
- Flux Specific Gravity
- Paste Flux Viscosity
- Visual
- Flux Spread
- Wetting Balance
- Fungus

Does it meet Telcordia (formerly Bellcore) GR-78-CORE requirements?

- Copper Mirror
- Halides by Silver Chromate
- SIR - Electromigration

1.3 Terms and Definitions This handbook uses terms and definitions common to the electronics manufacturing industry and defined in **IPC-T-50** *Terms and Definitions for Interconnecting and Packaging Electronic Circuits.*

1.4 Using Solder Paste Solder paste is the material that provides a wetted solder connection between solderable surfaces. To accomplish that end, it consists of:

- A flux vehicle to break down surface oxides on the surfaces to be wetted, and
- Solder alloy in the form of very small solder balls that will coalesce into a solder connection when reflowed.

To be usable, the solder paste must:

- Have reasonable working life under ambient temperature and humidity conditions.
- Have sufficient surface tension, called tack, to hold the component leads/terminations in position until and during reflow.
- Not contribute to the formation of undesirable solder balls/solder fines on the assembly.

Solderability, printability, stencil life, slump, tack, fume collection during reflow, formation of solder balls, tombstoning, cleaning, probing through solder paste residue after reflow, and solder paste shelf life will be discussed in the following sections.

2 APPLICABLE DOCUMENTS

The following specifications and standards form a part of this handbook / guideline to the extent specified herein. If a conflict of requirements or guidelines exists between IPC-HDBK-005 and the listed applicable documents, IPC-HDBK-005 **shall** take precedence.

2.1 IPC[1]

IPC-A-20/21 Standard Pitch Stencil Pattern for Slump

IPC-SC-60 Post Solder Solvent Cleaning Handbook

IPC-SA-61 Post Solder Semi-Aqueous Cleaning Handbook

IPC-AC-62 Aqueous Post Solder Cleaning Handbook

IPC-CH-65 Guidelines for Cleaning of Printed Boards and Assemblies

IPC-A-610 Acceptability of Electronic Assemblies

IPC-TM-650 Test Methods Manual[2]

2.2.14.3 Determination of Maximum Solder Powder Particle Size

2.4.34 Solder Paste Viscosity - T-Bar Spin Spindle Method (Applicable for 300,000 to 1,600,000 Centipoise)

2.4.34.1 Solder Paste Viscosity - T-Bar Spindle Method (Applicable at Less Than 300,000 Centipoise)

2.4.34.2 Solder Paste Viscosity - Spiral Pump Method (Applicable for 300,000 to 1,600,000 Centipoise)

2.4.34.3 Solder Paste Viscosity - Spiral Pump Method (Applicable at Less Than 300,000 Centipoise)

2.4.35 Solder Paste - Slump Test

2.4.43 Solder Paste - Solder Ball Test

2.4.44 Solder Paste - Tack Test

2.4.45 Solder Paste - Wetting Test

2.4.47 Flux Residue Dryness

2.2 Joint Industry Standards[3]

J-STD-001 Requirements for Soldered Electrical and Electronic Assemblies

J-STD-002 Solderability Tests for Component Leads, Terminations, Lugs, Terminals and Wires

J-STD-003 Solderability Tests for Printed Boards

J-STD-004 Requirements for Soldering Fluxes

J-STD-005 Requirements for Soldering Pastes

J-STD-006 Requirements for Electronic Grade Solder Alloys and Fluxes and Non-Fluxed Solid Solders

2.3 Military[4]

MIL-STD-203 Aircrew Station Controls and Displays: Location, Arrangement and Actuation

MIL-STD-883 Test Method Standard, Microcircuits

2.4 International Electrotechnical Commission Documents[5]

IEC 68-2-20 Environmental Testing. Part 2: Tests. Test T: Soldering

2.5 Telcordia[6]

GR-78-CORE Generic Requirements for the Physical Design and Manufacture of Telecommunications Products and Equipment

2.6 ASTM[7]

ASTM D 1210 Standard Test Method for Fineness of Dispersion of Pigment-Vehicle Systems by Hegman-Type Gage

3 SOLDERABILITY

This handbook will help the user understand solderability and wetting and their role in SMT soldering. Referenced test methods are generally accepted by the industry and are currently used to provide the data for product acceptability.

3.1 Theory of Wetting Wetting of solder to a metallic surface is driven by the thermodynamic surface energy of the substrate and liquid solder before and after the soldering process. In SMT soldering, the high surface energy of the substrate plays a key role in creating a wettable surface.

1. www.ipc.org
2. Current and revised IPC Test Methods are available on the IPC website (www.ipc.org/html/testmethods.htm).
3. www.ipc.org
4. http://assist.daps.dla.mil/online/start
5. www.iec.ch
6. www.telcordia.com
7. www.astm.org

For molten solder to form a strong solid state bond with a given metallic substrate, the following conditions must be fulfilled:

- Metal substrate and solder alloy should be compatible;
- Both the solder and the substrate must be free of oxides and foreign material and remain so during the reflow cycle; and
- Temperature must reach a sufficient level.

When solder paste reflows and wets a metal substrate, three components are present: molten solder metal, substrate metal (a solid) and flux.

As the molten solder wets the substrate, its area of spread is dependent on the conditions listed above. Depending on the solder alloy, good wetting may be indicated by a contact angle that is less than 75° (see Figure 3-1).

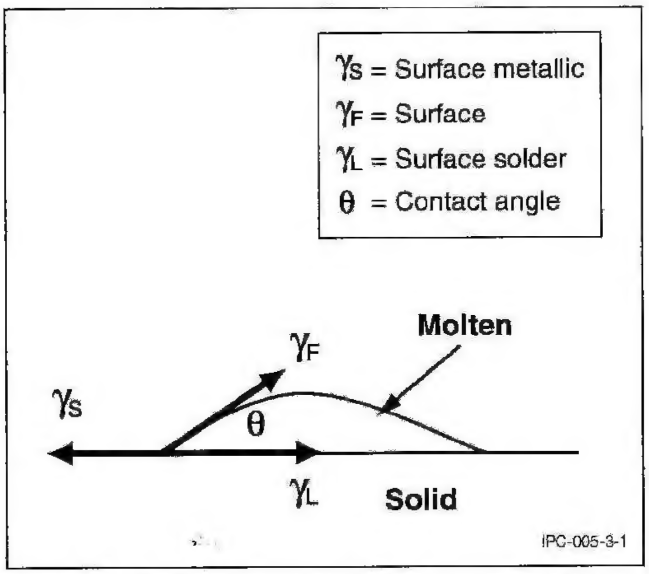

Figure 3-1 Surface Tension and Contact Angle of Molten Solder

The wetting is governed by the equation: $\gamma_s = \gamma_F \cos\theta + \gamma_{LS}$

In general, for wetting of the substrate by the liquid solder to occur, the surface free energy of the substrate (γ_s) must be larger than the sum of the surface tension of the molten solder metal (γ_{LS}) and surface tension of the flux (γ_F) times the cosine of the contact angle. In most cases, the surface energy of the substrate (γ_s) and the molten solder metal (γ_{LS}) is much larger in magnitude than γ_F. Their effects play a greater role in determining wetting. In other words, if the surface tension of the substrate is larger than that of the molten solder, the solder wets the base metal well. A metal free of surface oxide has a larger surface energy and thus, fluxing, or removal of oxide, improves wetting (γ_s increases).

Using the example of a Pb-Sn solder on a clean Cu substrate:

- Pure Pb does not wet a pure Cu substrate at typical soldering temperatures. This is due to a large interfacial tension between Pb and Cu; if Sn is added to the Pb, the wetting improves. The addition of the Sn reduces the interfacial tension between the solder alloy and the Cu substrate. Further addition of Sn to the Pb causes the surface tension of the molten solder to increase, and the contact angle of the Sn-Pb alloy to increase.
- These same principles of theoretical wetting of a substrate by molten solder apply to lead-free alloys also. The specific wetting angle of each alloy has to be considered in evaluating lead-free alternatives to each other and Sn-Pb alloys.

3.2 Degrees of Wetting Non-wetting between two metals occurs when the molten solder is unable to adhere to the base material. No attachment or bond is formed, the solder does not spread, and the base material is visible.

Dewetting is caused if the liquid to solid interface is not continuous, and areas exist were there is no adhesion. The solder spreads and then retracts, resulting in areas exhibiting uneven coverage with irregularly shaped solder mounds separated by a thin layer or film of solder. Some areas of the base material may be exposed.

Insufficient wetting is caused when the solder layer on the surface of the base material does not completely wet the entire surface. The layer is continuous, but only partially covering the surface of the base material.

Factors to be considered as causes of these conditions, and their effects are listed in Table 3-1.

3.3 Wetting Measurement Changes in the area coverage by the solder can be measured as adjustments are made to the controlled variables. These include temperature, time and materials. The angle of the fillet produced during soldering indicates the degree to which the base material was wet by the solder and can be measured from the cross section of the solder connection.

3.4 Factors Affecting Solderability During Reflow Soldering Process The factors that affect the solderability of a given substrate are:

- Land surface finish
- Component lead composition/finish
- Reflow profile
- Solder paste

3.4.1 Land Surface Finish Type, quality and thickness of surface finish on the PCB can greatly affect solderability. Coating thickness and quality both have a large impact on how the solder paste wets the land. A thin coating could be entirely consumed during the reflow process, especially if the boards are reflowed twice. The intermetallics formed can passivate very easily, and result in poor solderability. The quality of the coating is another factor. It should be

Table 3-1 Factors of Wetting and Their Effects

Factor	Effect
Uneven heating	During the reflow portion of the profile, the liquid solder will flow towards the areas of higher temperature. These areas may not be in the area of the required fillet.
Incompatibility of solder and substrate	The incompatibility of materials, metallization, surface finish or composition may not allow the formation of intermetallics thus prohibiting wetting.
Oxide formation	Oxides may occur on any of the metal surfaces involved in the forming of the fillet, either prior to or during reflow, and will inhibit the wetting of the solder through the formation of intervening oxides.
Presence of foreign non-metallic materials	Substrate manufacture, surface finishes and material handling are sources of contamination. All types of contamination can reduce the strength of a connection or prevent wetting.
Presence of additional thermal mass	Portions of the board with a high thermal mass will heat less quickly than the rest of the board and may not reflow even though the rest of the board does. Thermal load in the oven due to fixtures used in assembly process or other boards can cause cool spots in an oven and prevent complete and consistent reflow.
Surface or area of preferential metallization.	Lead finish, board finish and surface exposure will influence solder flow. Solder will wet to those surfaces which were cleaned by the flux, but difficult to clean surfaces, due to location or material, may be left unsoldered.

free of unwanted organic materials and be a uniform thickness.

3.4.1.1 Hot Air Solder Leveling (HASL) Hot Air Solder Leveling is the most common board finish. In general, it provides a good surface for the solder paste to wet the land. In addition, it rétains its solderability well. However, problematic thickness control which leads to a domed upper surface and lack of flatness can cause coplanarity, component skew, stencil smear and machine vision recognition issues.

3.4.1.2 Plating/Immersion Coating Electroplating, electroless plating, and immersion coatings are all alternative methods for depositing a metallic finish on a PCB. Noble metals such as Pd or Au are often used, especially on Ni surfaces. Gold and silver coatings that are too thin can cause dewetting and may be entirely consumed during long and multiple reflow exposures. The intermetallics formed can passivate easily, and will result in poor solderability.

3.4.1.3 Organic Solderability Preservatives (OSP) These protect the copper land surface with organic complexing agents such as benzamidazole or benzatriazole. OSPs provide a more level surface for soldering. However, they suffer both with un-assembled printed board shelf life and ease of solderability when compared to HASL when it comes to solderability because the protective monomolecular layer of organic molecules is so thin, it may not be robust enough. In some cases, it is easily removed by friction experienced during handling of the board. Multiple thermal excursions during storage or assembly can also degrade the coating, which may affect solderability. It is a characteristic of connections made over OSP coatings that wetting occurs only where the solder paste was placed. If the solder paste does not cover the entire land, it may not wet to the edge of the land.

3.4.2 Component Lead Composition/Finish Solderability of component leads are affected by the following factors:

- Base metal
- Coating composition
- Coating technique

3.4.2.1 Base Metal The base metal of the lead may consist of one or the following metals/alloys, each of which exhibits different wetting characteristics:

- Copper
- Copper alloys
- Alloy 42 (41-42.5%Ni, Balance Iron)
- Kovar® (29% Ni, 17% Cobalt, 53% Iron, 1% others)

3.4.2.2 Coating Composition Most component leads are coated with easily solderable metals to improve solderability. Sn and Sn-Pb coatings are commonly used. However, if additional requirements such as low contact resistance are required, coatings of Pt, Pd or Au may be applied.

3.4.2.2.1 Sn-Pb Coated Leads Sn-Pb coatings can be applied by electrodeposition or immersion. Sometimes an intermediate layer of Ni is applied between the copper substrate and the Sn-Pb coating to minimize formation of brittle Cu-Sn intermetallics on the lead.

3.4.2.2.2 Au Coated Leads While using Au plated leads, the cleanliness of the underlying material is very important, because the coating is generally very thin. Thus, it dissolves rapidly in the molten solder because of the high solubility of Au in Sn. If the underlying substrate is dirty or oxidized, it will dewet after the dissolution of the Au top layer into the molten solder. Finally, if all the Au is not dissolved, a layer of Au-Sn intermetallic can make the interface weak.

Au coatings are applied as thin as possible for a number of reasons:

- Cost
- Au forms an intermetallic with Sn that is very brittle
- By limiting the amount of Au available, this embrittlement can be minimized. It has been shown that more than 3% Au in the solder connection leads to unacceptable embrittlement of the connection. Also, dull-looking connections are often observed because of the Au-Sn intermetallic.

Thin Au coatings are generally very porous, and components that are stored for a long period of time can exhibit oxidation of the substrate material. This degrades the solderability of the leads.

3.4.2.2.3 Pd or Pt Plated Leads Pd or Pt leads may require a higher peak temperature and/or longer time above liquidus than normal Sn-Pb coated boards in order for the Pd to completely dissolve and expose the solderable Ni underneath. The finished connection also has a noticeable transition from the solder to the lead. Although the solder may show a higher contact angle on Ni than on Cu, the angle is acceptable as long as there is good wetting to the heel of the lead. Wetting forces on Ni are lower than on Cu so the force criteria used for copper substrates in the wetting balance solderability test should not be applied to Ni.

3.4.2.3 Lead Coating Processes Common component lead coating processes include the following:

- ***Electroplating*** This provides perfectly flat lead surfaces but it usually contains organic contaminants from the plating bath. Such organic contaminants can cause solderability problems.
- ***Electroplating and Fusing*** This process yields a surface that is as uniform as a plating surface and lesser organic contaminants.
- ***Hot Dipping*** Hot dipping process provides a nice coating of Sn or Sn-Pb but the coating thickness uniformity cannot be controlled.

3.4.3 Reflow The method of reflowing the solder connection greatly influences wetting, as discussed in the following:

- ***Inert or Reducing Atmosphere*** An inert or reducing atmosphere provides improved solderability by preventing or slowing atmospheric oxidation of the lead finishes and solder alloy.
- ***Peak Temperature*** The higher the peak temperature, the more the solder paste wets the land and wicks up to the lead, provided that the paste vehicle system remains active and fluid.
- ***Profile Length*** A longer time above liquidus normally results in better wetting as long as the paste flux is not consumed during the preheat and soak processes. The preheat ramp rate and/or soak time of the profile affects the total amount of time the flux has to act on the sub strate, lead and solder powder. If this time is too short, the flux does not have enough time to clean the surfaces to be soldered. However, if the time is too long, the flux may be used up before it has time to remove all of the oxidation. In addition, an extensive time above liquidus can lead to excessive intermetallic compound growth.

3.4.4 Solder Paste The composition of the solder paste has a large impact on wetting. Selection needs to consider flux type, metal alloy type, and the size of the solder powder.

3.4.4.1 Flux J-STD-004 is the standard used to classify fluxes. The activity of the flux used in the solder paste determines how well the paste will solder.

The flux used in the solder paste needs to be compatible with the alloys being used in the board finish, lead metallization, and solder alloy. For example:

- Metals such as Cu, Au, Ag, Pd, Sn and Sn Pb can be soldered using most available flux types.
- Cu alloys, Kovar, stainless steel and Zn are normally soldered with an inorganic flux.
- Aluminum alloys require a very specialized flux. These fluxes contain inorganic salts, such as chlorides and fluorides of zinc, tin, potassium and ammonium.

3.4.4.2 Solder Alloy The selection of a solder alloy to be used in solder paste is based on the substrate materials, final solder connection properties and operational temperature of the finished assembly. Eutectic alloys Sn63Pb37 and Sn62Pb36Ag2 are the most commonly used solder alloys. They have the following useful properties:

- Good wetting due to the affinity between tin and many other metals.
- The solder alloy itself has no brittle intermetallic compounds.
- Melting temperature of the eutectic alloy is below the temperature that would damage components present in common assemblies, yet above the service temperature of the completed assembly.

The composition of the solder alloy will also affect solderability. It has been shown that varying the composition of a Pb-Sn solder alloy affects the area of spread when applied to a clean copper surface.

Figure 3-2 shows that the addition of Sn to Pb (up to a certain composition) decreases the surface tension between the solder alloy and substrate material.

3.4.4.3 Powder Particle Size Smaller diameter solder powder means larger surface area that must be cleaned by

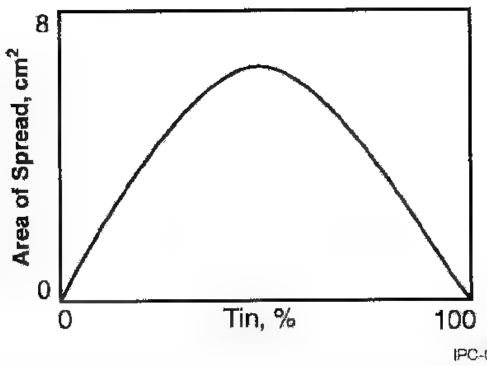

Figure 3-2 Effect of Alloy Composition on Area of Spread

the flux. This can result in reduced wetting if the amount of flux is insufficient to completely remove all of the oxidation from the solder powder.

3.4.5 Summary Wetting of a solder paste is dependent on the following factors: board surface finish, component metallization, reflow profile, solder alloy and flux.

3.5 Solderability Testing The purpose of solderability testing as it applies to solder paste involves analyzing the wetting ability of the paste. There are other factors that affect solderability such as the metal substrates, component leads and terminations. To evaluate these materials the user is referred to J-STD-002 and J-STD-003.

3.5.1 Solder Paste Wetting Tests The wetting ability of solder paste is determined using IPC-TM-650 Method 2.4.45 and the evaluation criteria as given in J-STD-005. This test evaluates the effective wetting of the solder paste on a solderable surface.

3.5.2 Solderability Test Documents The measurement techniques and systems that can be used for solderability evaluations are listed in J-STD-005, J-STD-002, and J-STD-003. Table 3-2 provides a listing of commonly used solderability test methods.

3.5.2.1 Special Solderability Test Methods The following are several other solderability test methods that users may find of interest:

- ***Sequential Electrochemical Reduction Analysis (SERA)*** This is a chronopotentiometric method that involves the reduction of surface oxides in sequence according to their electromotive reduction potentials. This method can determine the type of oxide on the surface and its thickness. While this is yet to be recognized in the industry as a standard solderability test tool it has shown to be of value in some instances.
- ***Ceramic Plate Test*** (A modification of IPC test method TM 2.4.43) The ceramic plate test is another test that can be used to evaluate the wetting of a solder paste. This test involves printing a brick of solder paste onto a non-wettable surface, such as a ceramic plate. Then a component with leads of known-good solderability, e.g., SOIC or QFP, is placed on the solder brick and reflowed. The wetting of the solder paste onto the component leads is then evaluated visually for evidence of de-wetting or non-wetting after reflow. This test should be performed with ramp and dwell temperatures and times that closely resemble the actual production process conditions. If there is evidence of de-wetting or non-wetting, this may indicate that there is a problem with either the solder paste or the processing conditions.

Table 3-2 Common Solderability Test Methods

Name of the Test	Application	References	Comments
Edge Dip Test	PCB, Lead	J-STD-003 MIL-STD-203 MIL-STD-883 IPC-S-803 IEC-68-2-20	Visual Rating; most common in the industry; lest expensive method.
Wetting Balance	Lead	J-STD-003 MIL-STD-883 IEC-68-2-20	Quantitative and graphs used, expensive.
Globule Test	Lead	J-STD-003 IEC-68-2-20	Numerical rating for flux, solder, etc.
Rotary Dip	PCB	J-STD-003	Visual Rating
Solder Float	PCB	J-STD-003	Visual Rating
Wave Solder	PCB	J-STD-003	Visual Rating
Spread Test	Solder Paste	J-STD-005	Visual and quantitative test

3.5.3 Solderability Test Method Selection This section lists some of the variables that should be considered when applying the tests listed in this guide. The results of each test are dependent on the interaction of these parameters. Controlling the following parameters is mandatory to produce reliable results:

- ***Method*** Acceptable test methods are listed in IPC-TM-650.
- ***Equipment*** Equipment to be used in testing is recommended in the applicable standards, other acceptable equipment may be acceptable in some instances. When using any equipment for evaluation purposes, it is recommended that the equipment is calibrated and a known or accepted standard is included in the evaluation. As with all testing, repeatability and reproducibility are to be considered prior to the acceptance of the data.
- ***Material*** This section may be considered the most critical area of an evaluation. Material interactions, preparations, storage, handling and reactivity may introduce uncontrolled variables into the evaluation. Specific issues are noted in the Environment section of this guide.

- ***Manpower*** Important consideration should be given to this area. The key to obtain accurate evaluation data is to understand how the following issues may affect an evaluation. Training, information, communication, cooperation and motivation can be sources of variation. Everyone involved with evaluations needs to address and understand these issues prior to the evaluation.
- ***Environment*** As mentioned previously the environment may contribute to the variability of results in an evaluation. Most methods will specify these conditions. When the conditions are not specified, recording them for the evaluation is suggested. These include temperature, humidity, atmosphere and sources of contamination, each of which may greatly affect chemical, mechanical and electrical testing.

4 PRINTING SOLDER PASTE

Printing is probably the most complex aspect of solder paste performance. The intention of this section of the handbook is to provide an understanding of the primary characteristics controlling solder paste deposition.

The printing process is governed by two solder paste parameters:

- Rheology
- Characteristics of the solvent system

The first is usually the most critical factor, and even today, little headway has been made in correlating the rheological properties of solder paste (which exhibits a combination of pseudoplastic, thixotropic and viscoelastic characteristics) to printing. The paste thixotropy is the most troublesome aspect, as the relaxation and retardation of the paste breakdown and build up in viscosity are frequently on a similar timescale to the print cycle.

Characteristics of the solvent system address both loss of solvent to the atmosphere (volatility), and absorption of moisture by the solder paste. This absorption is, of course, most critical to the performance of water-washable pastes, although even no clean pastes can exhibit concretion (hardening) effects in high-humidity conditions. Over the last five years, "enclosed print-head" printing innovations by some screen printer manufacturers have eliminated much of the variability associated with solvent loss and absorption.

The size of the solder powder also plays a role in finer pitch printing, since particles close in size to the aperture width either may or may not pass through, leading to increased variability.

4.1 Printing Needs The primary performance needs for a printing process are:

- A range of print speeds
- Capable of both fine and coarse pitch simultaneously
- Invariant with time

This last aspect is crucial for preventing the need for time-consuming, wasteful and possibly stencil-damaging paste clean-up and replenishment. The time-insensitivity has several aspects:

- ***Open Time*** Ideally, the print volume of even the smallest deposits should not vary with time, even after lengthy (30 minutes or more) pauses in the printing process. This aspect of solder paste printing is typically called "open time," "print after pause" or "abandon time."
- ***Slump*** The stencil may need wiping if the paste begins to drop to a low viscosity and begins to bleed onto the underside of the stencil. The need for wiping every few print cycles leads to variations in the print volume.
- ***Stencil-Life*** How long paste can be left on a stencil or screen is primarily a function of the solvent trends.

4.2 Printing Variables The solder paste manufacturer should provide recommended print parameters for benchmarking their materials. These may include:

- Recommended printer setup (squeegee angle, squeegee pressure, etc.)
- A range of print speeds
- Various lengths of abandon time
- Stencil design including aperture size, aperture shape, stencil thickness, etc.
- Squeegee type
- Underside stencil wipe frequency
- Humidity and temperature ranges

4.3 Primary Control Variables These are essentially the same as those for paste tack (see subsequent Section 5) - the 'ability of solder paste to retain components' (ASPaRC). During printing "raw" solder paste is exposed to the atmosphere. The major difference is the variation in paste rheology due to the material either rolling across the stencil or screen during the print cycle, or sitting motionless.

4.4 Printing This section describes the printability of a solder paste material using parameters recommended from the paste vendors. Room conditions should be within the range specified by the paste manufacturer. Minor parameters changes may be required for optimum printing.

4.4.1 Apparatus

- A solder paste printer
- A stencil, usually made of stainless steel, 0.15 mm [0.0059 in] thick, containing 0.4 mm [0.0157 in], 0.5 mm [0.0197 in] and 0.65 mm [0.0256 in] pitch QFP patterns (stencil to be designed and aperture dimensions defined)
- Flat panel substrate and/or matching boards (Note: For end users, the actual stencil and matching boards for production may be used, if desired.)

4.4.2 Procedures

a. Record the room conditions (temperature and humidity) at the beginning of the test and at each interval a print test is initiated to define the print environment conditions. Ideally the room is temperature and humidity controlled to about 25°C ± 2°C [77.0°F ± 3.6°F] and 50% RH ± 10% or within the range specified by the paste manufacturer or agreed upon between two parties. Exercise care to ensure that the paste is not positioned directly in the draft of any room or cabinet environmental control equipment.

b. Before testing, stabilize the solder paste to room temperature by either placing the sample in a constant temperature water bath or by leaving the sample overnight at room temperature. Stir the paste by hand for 30 seconds (about 30 turns) to obtain a uniform consistency. Record the temperature of the paste.

c. Set the print parameters of the printer as recommended by the solder paste vendor.

d. Transfer an appropriate amount of the material to be tested onto the stencil in front of the squeegee in the direction it prints. The amount should be adjusted to form approximately a 1-2 cm wide roll when printing is initiated. Record the paste temperature and room conditions (temperature and humidity).

e. Print continuously (on flat panel substrates), adjusting the parameters as needed. Wipe (cleaning the underside of the stencil) if necessary until the optimum printing performance is obtained.

f. When the print performance is at optimum level, replenish the solder paste to form an approximately a 1-2 cm wide roll. Thereafter, do not add more paste if performing the first part of step g (intervals printing). For continuous printing, add paste as required.

g. With these optimum print parameters, print 10 prints initially and then at 2 and 4 hours later (leaving paste on the stencil exposed to ambient conditions) after standing at the ambient temperature and humidity (other time intervals may be printed if desired). Record the temperature of the paste, the temperature and humidity in the printer chamber. This practice will enable one to deter mine the sensitivity of the paste to environmental conditions, the stencil-life and the paste's "Response to Printing Pause," at these conditions.

h. A similar observation can also be obtained from continuous printing. For the best Response-to-Printing-Pause characteristic, the print should be idle for about 60 minutes. The first few prints after idle time indicate the Response-to-Printing-Pause characteristics of the paste.

4.4.3 Evaluation

a. While printing, carefully examine the rolling of the paste as it moves across the stencil. Generally, a good printing paste rolls as the squeegee is shearing it across the stencil.

b. At the end of the print cycle, observe the release of the paste from the squeegee blade as it lifts up. Most of the paste should stay on the stencil when the squeegee lifts up ready for the next cycle to print. If the squeegee lift is small, a curtain of paste hanging at the end of the print cycle is acceptable.

c. Periodically after a few cycles and a few hours, observe the stencil apertures. There should be no clogging with paste, as evidenced by blinding of the apertures, and incomplete to no transfer of the paste onto the substrate.

d. All prints should be examined a few minutes after printing. Selective prints should be examined after 20 minutes standing at room conditions and there should not be much deviation from the first examination. The following print defects are to be examined for each print in each of the 16, 20 and 25 mil pitch QFP patterns: bridging, slumping, incomplete print, light prints, missing prints, wedge-shaped prints, print tailing, particles around prints and frequency of wipes.

e. Select a few sites and measure the height of the printed fillet. The paste height should be within 20% (or other set percent) of the stencil thickness or within the acceptable ranges.

f. If an automated solder paste inspection instrument is available, the pass/fail criteria should be set to 70 - 80% for passing.

g. After each print interval, the first few prints indicate the Response to Printing Pause characteristic of the paste. A good Response to Printing Pause solder paste material gives a full print with good definition immediately after idle time.

h. The length of time solder paste can be continuously printed in one shift with acceptable results, indicates the stencil-life of the paste.

i. Periodically, observe the paste on the stencil after extended exposure to the environment. There should be no drying or wetting of the paste due to humidity and/or temperature.

j. Record type of printer used (make and model), stencil (chem-etched, laser cut, etc.), and print parameters.

4.5 Solder Paste Print Quality Control Variables

4.5.1 PCB Variables

- ***Board Warpage and Planarity*** May impact stencil/board gasketing leading to inconsistent paste volume transfer and solder paste bleed-out under stencil.

- ***Solder Mask Height*** May impact stencil/board gasketing and solder paste volume transfer.
- ***Tooling Holes*** Impact board and stencil alignment.
- ***Plugged Vias*** Impact gasketing by raising stencil.
- ***Surface Finish*** HASL finish can affect gasketing of apertures and tenting (of the stencil) near large plated areas due to excessive domed surface of the coating. Poor flatness can also throw off vision alignment systems due to poor reflective characteristics.

4.5.2 Solder Paste Variables

- ***Powder Size and Distribution*** Excessive fines may increase viscosity resulting in poor aperture fill and release and overall poor print quality. Large particles may clog fine pitch apertures.
- ***Paste Tackiness*** Excessive paste tack may lead to squeegee blade hang-up and aperture clogging. Insufficient tack may result in placement defects.
- ***Paste Stencil Life*** Short stencil life requires printer parameter adjustments to compensate for paste variations and ultimately result in paste dry out or shear thinning.
- ***Paste Rheology*** Effects paste roll, aperture fill and release. Fine pitch printing requires shear sensitive material.

4.5.3 Stencil Variables

- ***Aperture Formation Method*** Whether chemically etched, laser cut, or electroformed, impacts paste release quality and deposition volume consistency.
- ***Stencil Thickness*** Impacts paste release and deposition height and volume. May lead to bridging or insufficient solder paste.
- ***Stretch During Stencil Assembly*** Impacts paste deposition accuracy.
- ***Trapezoidal Shape and Aperture Wall Smoothness*** Impacts paste release, deposition quality, bridging, and insufficient solder paste.
- ***Stencil Material*** Several options are available for stencil manufacture. Stainless steel, nickel, brass and Kapton all have unique characteristics that can enhance the print process.

4.5.4 Screen Printer Variables

- ***Squeegee Type*** Metal vs. Poly may affect paste volume. Poly blades can scoop paste resulting in decreased volume.
- ***Squeegee Speed*** May impact paste shear, paste roll characteristics, aperture fill and overall print deposition quality.
- ***Squeegee Pressure*** May affect gasketing, print deposition quantity, blade wear, and blade angle.
- ***Squeegee Angle*** Impacts paste roll characteristics and aperture fill quality.
- ***Underside Wipe Frequency*** Affects on aperture clogging, print quality, bleed-out, bridging, and solder balling will vary, depending on whether the wiping is done dry, wet or with vacuum.
- ***PCB Release Speed and Distance*** May impact aperture release efficiency, deposition quality and morphology, and clogging.

4.5.5 Environmental and Operator Variables

- ***Temperature*** High temp leads to paste slump. Low temp may cause overall poor printability.
- ***Humidity*** High leads to moisture pick-up, paste shear, paste slump. Low leads to paste dry out and poor printability.
- ***Paste Handling*** Temperature of paste at time of application may impact roll quality, fill, and release. Amount of pre-mixing before application can impact paste viscosity and printability.

5 ABILITY OF SOLDER PASTE TO RETAIN COMPONENTS (TACK)

The word "tack" has become synonymous with how well solder paste can hold a component in position during all operations prior to the formation of the reflowed connection. However, the variable commonly referred to as "tack" (the peak tack force established by use of IPC-TM-650, Method 2.4.44) has increasingly been seen as a poor indicator of a factor that may be referred to simply as the 'ability of solder paste to retain components' (ASPaRC). It is important to differentiate between these factors.

5.1 Background The IPC-TM-650, Method 2.4.44 "tack test" was developed at the end of the 1980s, at a time when component placement was a comparatively slow process. The primary reason that a solder paste deposit could not hold a termination in place was the rapid upward movement of the placement head. The test method chosen was therefore one that gives as a measure of tack, a peak tension force, as the result of a rate-controlled extension.

5.2 Tack Test Shortcomings The current tack test makes several assumptions, the most important of these being:

- The peak extensional force during probe removal is a valid indication of ASPaRC in an SMT production process (see Figure 5-1).
- The test uses a 0.25 mm [0.0098 in] thick deposit, which is unrealistic in today's manufacturing environment.
- The test uses a 300-gram insertion force, which is unrealistically high.
- The test is extremely slow, favoring viscous dissipation of the applied probe-force, rather than the elastic "bounce" response sometimes seen during ultra-fast placement.

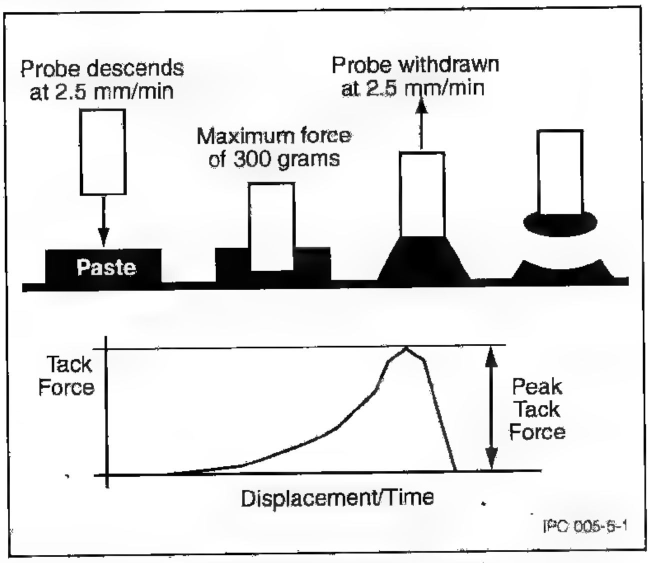

Figure 5-1 The Extensional Nature of "Tack"

- The "tack life" data can be reported in three ways, which eliminates the ability to compare data from different manufacturers.

5.3 Tack Life Although ASPaRC is used as a general term, it is probably best to use the name "tack life" to describe the length of time after printing, during which a paste is able to retain the placed parts when subjected to normal operational procedures. This may include high-speed pick-and-place equipment ("chip-shooters"), the manual handling and population of a board, and/or the movement of boards on conveyors or in board-stackers. Changes in solder paste rheology are almost always deleterious to the performance of the paste. The total useful working-life of a paste can be seen to be a function of four basic "life" parameters that we will call A, B, C and D; see Figure 5-2 and Table 5-1. This means that if a paste has been stored incorrectly, or has spent many hours exposed to adverse conditions on a stencil, then there will be a resultant shortening of the tack life of the paste.

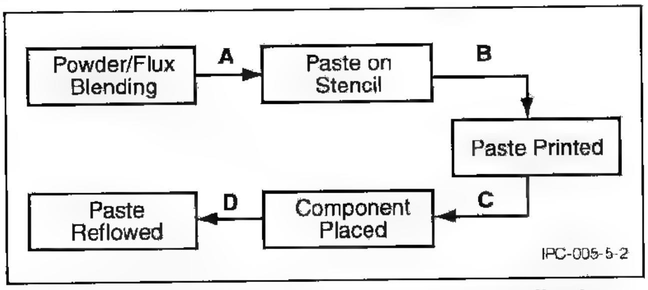

Figure 5-2 The Interaction of Various Factors Affecting Solder Paste 'Working Life'

"Storage" (A) and "Time on Stencil" (B) are discussed in other sections of this document. "Open Tack" (C) and "Retention Time" (D) however, are germane to this section, and can be pooled together under the general term "tack life." Therefore, there are two kinds of tack life:

Table 5-1 Main Control Variables Affecting Solder Paste Working Life

	Parameter	Control Variables
A	Storage	Time, temperature, amount of stirring, packaging
B	Time on Stencil	Time, temperature, humidity, gas velocity over paste, print speed, prints per hour
C	Open Tack	Time, temperature, humidity, gas velocity over paste, deposit thickness
D	Retention Time	Time, temperature, humidity, gas velocity over paste, deposit thickness, reflow preheat

- ***Open Tack (C)*** The length of time after printing on a circuit board, during which parts can still be placed in the solder paste deposits without either falling off, or causing an 'open connection' on subsequent reflow.
- ***Retention-Life (D)*** The length of time after placement during which a part remains in place prior to formation of the final reflowed connection.

D is usually greater than C: that is, most solder pastes retain the part, once placed, for a much longer period than the time over which they allow parts to be placed. The adhesion force of some pastes may even increase as the deposit dries around the leads. In addition, most operations have a longer time between printing and placing than between placing and reflow. Therefore, open tack is the more important factor.

5.4 Increased Tack Strategies Methods used to increase tack (as tested to IPC-TM-650, Method 2.4.44) may not necessarily give increases in ASPaRC due to the discrepancy between the two types of parameter.

5.5 Increased Tack Life Strategies Solder paste formulators can use a variety of means to give an increased solder paste life. The following lists potential drawbacks:

- ***Change in Metal Loading*** Reduction of the metal loading may give a small increase in the open-tack of the solder paste, but there may also be a tradeoff in slump.
- ***Low Volatility Solvents*** High boiling point (low vapor pressure) solvents may also be used as a means of improving the open-tack life. However, this can sometimes increase the amount of flux residue that may accumulate in the reflow oven as either solids (causing increasingly frequent maintenance) or liquids that may drip onto the finished assemblies. The reflow oven manufacturers have responded by devising a variety of different means of removing flux vapors from the oven.

5.6 Control Variables The initial ASPaRC is a function primarily of the solder paste, and most particularly, the flux formulation. However, several variables impact on the tack life (most particularly on the Open-Tack). The most important of these are:

- ***Temperature*** Higher temperatures evaporate solvents faster, and also increase the rate of concretion reactions, in the case of high humidity.
- ***Air Flow*** The velocity of gas flowing over solder paste is another factor governing the solvent evaporation rate.
- ***Humidity*** High humidity may increase the rate at which water vapor is absorbed by the solder paste. In the case of water-soluble paste, the deposit may slump appreciably, as the viscosity of the flux decreases. No-clean pastes, on the other hand, may harden. In both cases, there is a significant reduction in the tack life.
- ***Deposit Size*** Thicker and wider paste deposits lose solvent more slowly, and may crust-over in high humidity, to trap tackier solder paste underneath. Smaller, thinner deposits dry out quickly.

5.7 Improved Testing

5.7.1 Production Testing Every solder paste formulation can behave differently under different environmental conditions. From a user's point of view, it is much more valuable to have a test performed under actual manufacturing conditions rather than under an idealized laboratory environment. Formulation "X" may appear to have low tack under lab conditions, but may outperform other formulations in the end users' environment. It would not serve the user to discount formulation "X" based on published data under a specific, but irrelevant, set of conditions.

5.7.2 User Test Guidelines Any new test should try to simulate production conditions. Factors include deposit thickness, placement force, the above-mentioned environmental concerns, part/land geometries and magnitude of external forces encountered.

The ideal test would cover both parts (open-life, C, and retention-life, D) of the tack life question. The test would determine the amount of force required to shear off a part. The test would also determine the amount of time a paste would retain a part above that shear force.

If a relationship between part size, weight, and footprint to the shear force could be established, and the shear stress exhibited by the most violent pick and place machine determined, then one standardized test may work.

A simple test that is often used is to print a board, and every 30 minutes attempt to manually place a PLCC J-lead component on the board. A 10-gram [0.35-ounce] weight is placed briefly on top of the part to try to standardize the force of placement. The board is then slowly rotated to see if the part holds. The time after which no parts fall off the PCB is established as the functional tack time. J-leaded parts are a good test part due to their high weight to contact area. Variables would include, the actual placement force, the speed of rotation of the board, and the change in momentum when stopped. Since this test can be done on the manufacturing floor, it becomes a functional test, only applicable to those particular conditions.

The above test does not address an increasingly common failure mode: the skewing or sliding of components when subjected to high-speed pick-and-place equipment. The rapid x-y axis acceleration and deceleration during high-speed placement is the main area of concern for solder paste users. A simple but effective test that can simulate the shear forces has been proposed. The first part of the test incorporates a tilt test similar to the test described above. A number of metal electrode face bonding (MELFs) are placed, and the number that falls off is recorded. The second test is a drop test, where the board is dropped from a fixed distance on its edge to simulate the deceleration of a board during rapid board handling.

6 SLUMP

This section provides guidance on evaluating relative slump performance of solder pastes. Test procedures are designed around standard SMT production conditions with an emphasis on comparisons between solder paste types or lots of the same solder paste. Slump, in the context of SMT production, describes a situation where the viscosity of a solder paste deposit is not high enough to resist the force of gravity. When slump occurs, the solder paste deposit droops into a rounded form with a broad base. Severe slump can lead to wet bridging of solder paste deposits between lands. Slump impacts processes in several ways: solder bridges are more likely to form during reflow, depleted connections due to wicking is more likely, solder balling and solder beading increases, and opens could increase. Solder paste slump should be broken into several types:

- ***Fresh vs. Conditioned Paste*** Solder paste for SMT is designed to decrease in viscosity as it is sheared and recovers in viscosity after the shear is removed. This rheological property is defined as thixotropy. Because of this thixotropic property, fresh paste intrinsically has a higher viscosity than conditioned, or sheared, paste. If the viscosity of the solder paste is unable to recover after the shear force is removed, the potential for slumping is increased.
- ***Cold vs. Hot Slump*** Understanding where the paste slump takes place is essential in determining the proper solution to this problem. Slumping that occurs after the screen printer and before the reflow oven is typically termed "cold slump." If slumping occurs in the reflow oven as a result of the increasing temperatures, this phenomenon is defined as "hot slump."

Note: IPC has a Standard Pitch Stencil Pattern for Slump, IPC A-20/21, which ideally accompanies production testing.

6.1 Slump Testing Apparatus

- A Solder Paste Printer
- A stainless steel, 0.15 mm [0.0059 in] thick stencil containing 0.4 mm [0.0157 in], 0.5 mm [0.0197 in] and 0.65 mm [0.0256 in] pitch QFP patterns with the corresponding flat panel substrate. Production boards with the fine pitch locations would be appropriate for this test.

6.2 Test Procedure The room conditions (temperature and humidity) should be recorded at the beginning of the test and at each interval, a print test is initiated to define the print environment conditions. Ideally, the room temperature and humidity is controlled to about 25°C ± 2°C [77.0°F ± 3.6°F] and 50% RH ± 10% or within the range specified by the paste manufacturer or agreed upon between two parties.

a. Before testing, the solder paste sample shall be stabilized to room temperature by either placing the sample in a constant temperature water bath or by leaving the sample overnight at room temperature. Stir the paste by hand for 30 seconds (about 30 turns) to obtain a uniform consistency. Record the temperature of the paste.

b. Adjust the parameters of the printer to vendor recommended settings or settings that have previously shown good results.

c. Transfer about 350 - 400 g of the material to be tested on to the stencil in front of the squeegee in the direction it prints. The amount should be adjusted to form a 1-2 cm wide roll when printing is initiated. Record the paste temperature and room conditions (temperature and humidity).

d. Use the same stencil and board lot for all testing. Differences in etching during board and stencil fabrication can affect results.

e. Print continuously (on flat panel substrates), adjusting the parameters as needed to obtain the optimum printing performance. Wiping the underside of the stencil may be necessary. Record these settings and the final wipe frequency. This optimization prevents misinterpretation of poor print quality as slumping.

f. When the print performance is at optimum level, clean the stencil and blades. Begin the test prints with fresh (non-printed) but conditioned (room temperature) solder paste. Add enough solder paste to form a 1-2 cm wide roll.

g. Record the temperature and humidity in the printer chamber.

h. Turn off the stencil auto-wipe feature. With the optimum print parameters, print 10 boards and set them aside for evaluation (these are designated the *fresh* boards). Record the time.

i. Set knead cycle to 100, run the knead cycle, eject and clean the board and bottom of the stencil, print 10 test boards and set aside for evaluation (these will be designated the *knead-100* boards). Record the time.

j. Set knead cycle to maximum (usually 999), run the knead cycle, eject and clean the board and bottom of the stencil, print 10 test boards and set aside for evaluation (these will be designated the *knead-999* boards). Record the time.

6.3 Evaluation

6.3.1 Cold Slump After initial inspection of the 30 boards set aside for evaluation, place 5 from the *fresh, knead-100* and *knead-999* boards in a safe storage place. Inspect these boards every hour for four hours. These results determine if the solder paste is susceptible to cold slump.

6.3.2 Hot Slump Place the other 5 from the *fresh, knead-100* and *knead-999* boards in a location of known elevated temperature. Hot plates and ovens set to 30-100°C is recommended. Warm outdoor temperatures or setting boards above reflow ovens may also be acceptable. Inspect these boards every hour for four hours. These results determine if the solder paste is susceptible to hot slump.

6.3.3 Slump Performance In most cases, visual inspection is necessary to evaluate slump performance. Most automated inspection systems do not discern slumping from other potential defects nor do they assess the severity of slumping. Because subjective visual inspection is required, a control material should be used for direct comparison.

6.3.4 Visual Inspection Visually inspect all fine-pitch lands for the four primary indications of slump. Note the severity, frequency, and location of each.

- ***Rounding*** Rounding of the paste deposit indicates but does not guarantee that the paste will slump. Rounding is the softening of the sharp edges around the top of the paste deposit. Note that rounding will likely occur with most pastes when stored at the elevated temperatures. Surface tension of the flux may keep rounding from translating into actual slump behavior. Excessive peaking of the paste deposit may mask rounding and should be discounted.
- ***Flux Bleed-out and Flux Bridges*** These also indicate, but do not guarantee, that the paste will slump. Flux bleed-out occurs when the paste deposit is intact but flux appears to bubble or flow to the surface of the deposit and pool onto the land or mask. Often the deposit becomes shinier with time. Extensive flux bleed-out may lead to flux bridges between lands. The key characteristic that distinguishes this phenomenon from paste slump is that no powder is found in the pooled flux. If the powder does not move off the land, no defect occurs.

- ***Spreading*** Spreading of paste off of the land and onto the board or solder mask is true paste slump behavior and could lead to defects.
- ***Wet bridging*** is the most severe consequence of paste slump. Wet bridges form when the paste connects lands together.

6.4 Differentiation Differentiation between material performance can be achieved not only by assessing if the phenomenon described above occurred but also the extent and location of each. A scanned or photocopy image of the test board should be provided to inspectors so they can note the location of the bridges. In addition, they should note if the bridges occur on lands whose length is parallel or perpendicular to the squeegee blade length. Most pastes bridge more often on lands that are perpendicular to the squeegee blade length.

6.5 Major Control Variables The following are factors that will likely affect the frequency of slumping occurrences:

- ***Solder Paste Thixotropy*** Solder paste is a unique material in that it requires a low viscosity for good printing characteristics and a high viscosity to maintain a well-defined print shape and resist slumping. Thixotropy is the fluid flow characteristic that allows solder paste to achieve both the high and low viscosity in one product formulation. In the printer, the squeegee blade stroke induces shear on the solder paste. The shearing force decreases the paste viscosity and allows it to flow into and completely fill the apertures. The solder paste should then recover back to a higher viscosity that resists slump. A solder paste with a low thixotropy needs to be formulated to have a lower viscosity for acceptable printing characteristics. This combination of low thixotropy and low viscosity leads to an increased probability of slumping.
- ***Solder Paste Metal Load*** Increased metal loading of a solder paste has been shown to reduce the occurrence of hot slump dramatically. Solder paste consists of metal powder and flux. As the temperature of a solder paste deposit increases, the flux constituents of the paste decrease in viscosity. The powder constituents do not decrease in viscosity until they become liquidus and, at that point, the surface tension and wetting forces are high enough to resist slumping. Therefore, the more flux (or lower metal load) in a solder paste, the greater is the potential for the viscosity to decrease enough to cause slumping at elevated temperatures.
- ***Solder Paste Particle Size*** For typical SMT operations, Type 3 (-325/+500 mesh) powder is standard. A finer mesh solder paste can actually reduce the potential for slumping because it increases the viscosity. However, these smaller particle sizes can lead other problems, such as increased solder balling and poor wetting.
- ***Temperature and Rate of Temperature Change*** As the temperature of a solder paste deposit increases (as it is inside the reflow oven), two things begin taking place. First, most materials with a fixed composition, including solder paste, will decrease in viscosity (thermal agitation effect) in a function that is independent of time. This decrease in viscosity with increasing temperatures would normally lead to slumping. However, increasing temperatures also leads to a loss of solvents, which are the low viscosity portion of solder paste. Solvent volatilization results in increasing viscosity of solder paste, but requires both elevated temperatures and time to complete the phase change from liquid to vapor. Using a slow ramp-up rate in the reflow oven, less than 1 - 1.5°C/sec [1.8 - 2.70°F/sec], allows the solvent volatilization effect to counter the decrease in viscosity caused by the thermal agitation effect and reduces the potential for hot slumping to occur.
- ***Humidity and Humidity Resistance*** Moisture absorption into a solder paste leads to an increased potential for slumping, as well as several other defects. The amount of moisture absorbed into a solder paste is dependent on several variables. A high humidity means that there is more moisture in the air and more moisture available to be absorbed into the solder paste. Solder pastes are typically designed to be used at a relative humidity of 30 - 50%. The ability of a solder paste to perform well at a higher humidity is dependent on the hygroscopic nature of the specific formulation. For example, water-soluble solder pastes are typically more hygroscopic than no-clean solder paste fluxes and less tolerant of high humidity levels. In addition, using a solder paste that is significantly below the temperature within the screen printer can lead to moisture condensation on the solder paste and result in a high level of moisture absorption and increased slumping.

Note: Viscosity was not listed as a control variable. The viscosity of a solder paste, as tested by J-STD-005 and listed on most product data sheets, cannot be used to evaluate the potential for solder paste slump. Viscosity plays the primary role in the slumping phenomenon, but should not be listed as a control variable because it is dependent on all of the factors listed above.

7 SOLDER PASTE FUME COLLECTION AND DISPOSAL

This section provides guidance on evaluating the compatibility of a solder paste with the reflow oven and environmental considerations.

7.1 Background All solder pastes contain ingredients that volatilize or evaporate during reflow. These vapors contaminate the atmosphere inside the reflow oven and exhaust system, and may accumulate as deposits in the reflow oven or in the environment. These deposits may

contaminate the products, deteriorate the equipment performance, and may be hazardous to personnel. The solder paste fumes and deposits need to be controlled, collected, and disposed of in compliance with all applicable safety and environmental regulations.

There are several considerations that solder paste users need to address in evaluating a solder paste. These include the collection of the solder flux fumes and deposits in any flux collection system or exhaust system on the reflow oven, the toxicity of the collected materials, and the frequency and costs of the required system maintenance.

7.2 Flux Vapor Collection The volatile ingredients of the solder paste come off the product at temperatures that can be determined by Thermo-Gravimetric Analysis (TGA) of the paste. The TGA provides a graph of weight loss versus temperature as the solder paste is heated. All the original solder paste that does not remain on the product as reflowed solder and flux residue ends up in the reflow oven atmosphere. These fumes and deposits should be collected in the reflow oven or its exhaust system by an appropriate flux collection system. The fumes of different solder pastes collect as vapors, liquids, and solids (tars) depending on their ingredients and temperatures.

7.3 Residue Toxicity The collected fumes and deposits may contain toxic or hazardous ingredients in higher concentrations than in the original solder paste. The chemistry of these residuals may also have been altered by reflow in an oxidizing or inert atmosphere. In addition, the accumulated solder paste residuals may be combined with the fumes of Organic Solderability Preservatives (OSP) and other volatile compounds on the product. As a result, the collected residues may be hazardous or toxic. Appropriate care must be taken to protect the safety of operations and maintenance personnel.

7.4 Collection System Maintenance The maintenance of the solder reflow oven, its flux collection and cooling systems, and any exhaust system, can be significantly changed by the characteristics of the solder paste used. Both the quantity of the solder paste volatiles and the physical characteristics should be considered to determine the effect on maintenance costs. All collected solder paste fumes and deposits need to be treated and disposed of in compliance with all applicable environmental regulations.

8 SOLDER BALLS

While a number of factors can cause formation of solder balls, solder beading and splatter during the reflow process, the introduction of no-clean processes in electronic manufacturing has given rise to greater levels of solder balling. Solder balling appears as one or more spheres of solder separated from the main body of the solder connection, with a typical diameter of around 0.5 mm [0.020 in] or less. When carrying out a post-reflow cleaning of board assemblies, solder balls do not pose any significant threat to the reliability of electronic assemblies because they are washed away during the cleaning operation. However, no-clean flux residues are not removed from the assembly after soldering, and thus any solder balls that remain on the board, are usually entrapped in flux.

IPC-A-610 classifies solder balls as defects for all classes of assembly if they are:

- Mobile (not entrapped in flux), and so can cause shorts between adjacent lands or copper conductor traces on PCB assemblies, and/or
- Violate minimum electrical clearance.

Therefore, it is imperative that adequate measures are taken to prevent their occurrence. With the emergence of fine-pitch technologies such as 0402 and 0201 passive components and chip-scale packages and direct chip attach (CSP, DCA), it is becoming increasingly important that solder balls are eliminated during the assembly process.

8.1 Solder Ball Tests The current IPC solder balling test method (IPC-TM-650, Method 2.4.43) calls for the use of different stencils depending on particle size:

- For paste containing powders sizes 1-4: a 0.2 mm [0.0079 in] thick stencil with at least three circular apertures 6.5 mm [0.256 in] in diameter.
- For paste containing powder sizes 5-6: a 0.1 mm [0.0039 in] thick stencil with at least three circular apertures 1.5 mm [0.059 in] in diameter.

Some specimens are placed on a hot plate and immediately reflowed, while others are stored at room temperature at about 50% RH for 4 hours. These latter samples may be a source of spatter during hotplate testing.

The following points are criticisms of current test methods for assessing solder balls:

- It is qualitative and hence results are operator-dependent.
- The deposit dimensions are uncharacteristic of standard solder paste deposit dimensions.
- The lack of a preheat or ramp means that the test is easy to pass, unless spitting (solder spatter) occurs.

This latter is exacerbated by the rapid heating, particularly in the case of water-soluble pastes.

8.2 Forms of Solder Balling As has been shown above, there are several problems with the current solder ball test because it assesses only one of the types of solder balling usually seen. Actually, there are four primary mechanisms responsible for solder ball formation during reflow:

- Solder Beading
- Solder Paste Segregation

- Solder Spatter
- Lack of Coalescence

8.2.1 Solder Beads Solder beads (also known as 'capillary balls,' 'side balls' or 'squeeze balls') are seen beside discrete components with low standoffs and large lands with minimal separation, (usually chip capacitors or resistors). Solder beading is a function of a variety of factors, including the volume of solder paste printed, the reflow profile and component placement process; see Figure 8-1.

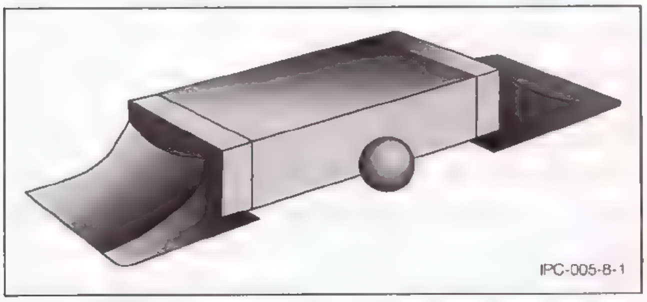

Figure 8-1 Solder Beading

Solder beading is the result of solder paste squeezing out under the component body during part placement. The solder trapped under the component cannot wet either the body of the chip or the PCB material. Surface tension works to minimize the surface area of the molten solder, forcing it out from under the component. A bead of solder therefore forms beside the component.

Paste squeeze-out can be minimized by changing stencil aperture design. If only the metallized terminations of the chip components are sitting on solder after component placement, all the paste should coalesce to the component terminations. This can be accomplished by reducing printed solder paste deposits.

8.2.2 Solder Paste Segregation Similar to solder beading, segregation involves a blob of solder paste becoming separated from the main deposit by either cold slump, or hot slump (that is, solder paste spread as the paste is heated). Gravity forces the solder out onto the surface of non-solderable areas (for example, solder mask) and solder balls form that are larger than the individual solder particles initially present in the paste. Solder paste can also bleed out from the bottom of the stencil and transfer to the solder mask on the PCB and form solder balls during the subsequent reflow process. Segregation is therefore a function of stencil cleanliness, cold and hot slump, paste deposit thickness, and other factors.

8.2.3 Solder Spatter (Solder Splash) Solder paste spattering occurs during reflow, and is characterized by finding the solder balls far away from the lands where the paste was deposited. The solder balls caused by spattering are generally in the order of the size of the solder powder in the un-reflowed solder paste.

Solder spatter may also turn up on gold/nickel surfaces, where it is usually called "solder splash." Depending on the thickness of the gold, solder splash may appear as a dull gray patch of solder (containing large amounts of tin/gold intermetallics) on gold edge-connectors (Figure 8-2-left), or as a shiny metal 'fried egg' on thin gold surfaces (Figure 8-2- right).

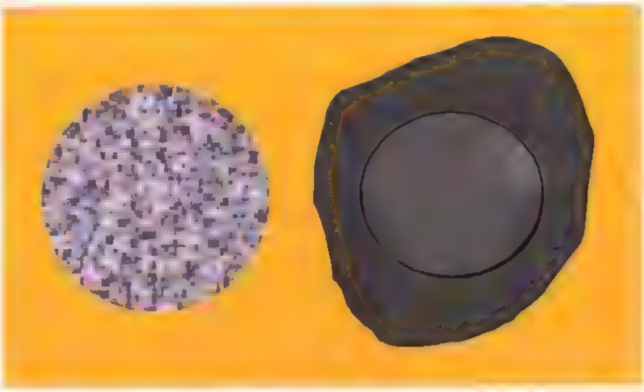

Figure 8-2 Solder Spatter

The solder spatter mechanism is not fully understood, but it can be caused by rapid heating of the solder paste, which causes the evaporation of solvent present in the solder paste flux during reflow. The vapor pressure associated with this process expels the solder paste from the main body of solder paste, which is thus unable to coalesce with the solder connection.

Solder spatter may therefore be associated with rapid volatilization of solvent or water, rapid outgassing of board-plating chemicals, or even low oxygen levels in reflow. Board storage and board quality are also known to be key parameters in eliminating this problem. Oven cleanliness is also an important factor.

8.2.4 Lack of Coalescence Solder balling, as tested by the IPC-TM-650, Method 2.4.43, and not caused by the above factors, is usually seen as one or more balls of solder powder failing to coalesce with a reflowed area of solder paste. In the worst case, this can appear as "fish egg" solder balls, illustrated as solder fines in IPC-A-610, where only part of the deposit reflows to form a connection.

8.2.5 Formulation Strategies To reduce or eliminate solder balling, as tested by IPC-TM-650, Method 2.4.43, solder paste formulators have several options available to them, in addition to overall formula optimization. The following lists potential drawbacks:

- ***Increase Flux Solids Content*** This can cause viscosity-stability problems, or cosmetic issues.
- ***Increase Levels of Activator*** This may cause crystallization in the flux, or lead to copper mirror reclassification and perhaps post-reflow problems with SIR or electromigration.

- ***Change Activator Type*** This can cause the same problems as with increasing the levels of activator.
- ***Improve Powder Quality*** Oxide on powder, and particularly the amount of finer powder (perhaps less than 20 µm [0.79 mil] in diameter) may be a factor in determining the extent of solder balling.

8.2.6 Process Strategies The process can also influence formation of solder balls.

- ***Change Profile*** A change in temperature profile to reduce solder balling and beading should be done in accordance with paste manufacturer's recommended specification.
- ***Change Atmosphere*** As solid-content, or activator levels, are reduced, it may become necessary to recommend reflow in an atmosphere containing less than the standard 21% oxygen (air).
- ***Control Variables*** The following conditions help determine the extent of solder balling caused by "lack of coalescence."
- ***Preheat*** Higher temperatures, and longer times at those temperatures, volatilize activators rapidly. Most organic materials have a measurable vapor pressure over 150°C [302.0°F]. Also, the oxidation of metal surfaces (solder powder and land/component terminations, alike) is increased by longer times at high temperatures. Lowering the preheat temperature and shortening the preheat time allows more paste vehicle to remain liquid into the reflow portion of the profile. This tends to reduce the formation of solder balls.
- ***Atmosphere Type*** Reduced oxygen environments (i.e., nitrogen) can potentially reduce or eliminate certain soldering defects such as solderballing. Nitrogen improves coalescence without increasing paste activation. Additionally, nitrogen offers an added benefit by eliminating reoxidation after the initial surface activation has been completed.
- ***Deposit Size*** Thicker and wider paste deposits lose activator more slowly than small, thin ones, due to their lower surface area/volume ratio.

8.3 Improved Testing Any new test developed should try to simulate production conditions. Factors to be considered in any new test include:

- Deposit size (thickness, width and depth), shape and orientation on substrate
- Storage conditions (temperature, %RH, local air velocity) and time of storage of prints
- Reflow profile (length/time of preheat) and atmosphere
- Heating method (IR / forced-convection / vapor-phase)
- Surface of substrate (i.e., solder mask and surface finish of metallization)

9 TOMBSTONING

One of the common defects seen in surface mount assembly with small discrete passives is referred to as "tombstoning," where the angle of the component-to-board-surface is close to 90°. This defect may also be referred to as "draw bridging" or "floating tombstone" when a column of solder extends from the bottom of the raised end to the top of the land and the component-to-board-surface angle is typically less than 45°.

Tombstoning occurs when a small-chip component flips up during reflow as shown in Figure 9-1. The solder on one of the lands reflows before the solder on the other land, or perhaps there is a large solder quantity difference so that surface tension on one end overcomes surface tension on the other. After the solder on the first land melts, the solder wets the component metallization and it pulls the component towards the land and the component flips up and stands vertically in the solder. The solder on the opposite land reflows after the component has lifted off the first land, subsequently contact to the second land is not made. This defect can also occur such that the component is not lifted fully into the air. In some cases the component may just slide or skew or be lifted up just enough that it rides on top of the reflowed solder, making an electrical connection, as in Figure 9-2. This defect would not appear in ICT or electrical testing, but may result in a field failure.

Figure 9-1 Tombstoned Component

9.1 Tombstone Causes Numerous papers have been presented at industry conferences on causes of tombstoning. Contributing factors include land design, reflow profile, placement registration, reflow oven and atmosphere, solder paste volume and solder paste characteristics.

Based on industry studies, the major contributing factor was found to be the PCB land footprint design. The second greatest factor was related to PCB loading into the reflow oven. It was found that loading the board or panel into the

Figure 9-2 Floating Tombstone or Draw Bridging Defect

reflow oven with the edge of the card containing the highest density of copper first caused these areas to begin heating before less dense areas. This minimized the temperature differences between lands for a given component. Numerous other papers have been published related to use of land design and geometry to help control this problem.

9.2 Minimizing Defects In the last several years there have been numerous papers detailing methods and solutions for minimizing and eliminating tombstoning defects. Work done by Adriance and Schake (see Appendix A) using a design of experiments (DOE), explored the effects of land geometry, land orientation, flux chemistry and processing atmosphere on reducing tombstoning of 0201 components. This work resulted in the following recommendations:

- Optimum land dimensions are 0.015 or 0.018 inches width and 0.012 inches length
- Spacing between lands should be 0.009 inches
- Components reflowed perpendicular to the application of heat experienced a greater degree of tombstoning
- Water-soluble paste in air and low-solids paste in nitrogen, produced more tombstoning defects than low-solids paste in air

Finally, solder paste can also play a role. In addition to the reduction in defects in going from water-soluble solder paste to low-residue, some studies have shown, that within types, there can be differences in solder pastes, as reported by Ganeshan (see Appendix B), and there are even solder pastes formulations designed to minimize tombstoning defects as reported by Toleno (see Appendix C). These papers are provided as Appendices B and C. These formulations typically work by using a blended alloy. This blended alloy contains solder particles of alloys that melt at two different temperatures (e.g., Sn63 @ 183°C [361.4°F] and Sn62 @ 179°C [354.2°F]). This mixture with the proper ratio and particle sizes produce a solder paste that begins to melt at 179°C [354.2°F], but not fully, such that the component is "tacked" down. Then the bulk of the solder paste reflows at 183°C [361.4°F], wetting the termination and the land, leading to a good solder connection. The only drawback is that the solder connections do appear a bit dull due to the fact that the material has a plastic range.

10 CLEANABILITY OF SOLDER PASTES AND RESIDUES

There are three cleaning issues to be addressed by solder paste users:

- Cleaning of stencils to remove solder paste left behind after the printing process is complete or when the stencil has become clogged (solder paste misprints). Operators should also clean the stencils prior to inspection for damage or prior to test printing or between prints to increase print quality.
- Cleaning of PCBs to remove solder paste misprints. At this point the paste has not been reflowed, so it is very similar to removing solder paste from stencils.
- Removal of post-reflow paste flux residues, which may be part of the rework process, or as a desired post-reflow process.

An IPC committee is developing a stencil cleaning handbook that will provide support for the first two issues. The third issue, removal of post-reflow solder paste residues by various methods, is detailed in the IPC cleaning handbooks IPC-SC-60, IPC-SA-61, IPC-AC-62, IPC-CH-65.

10.1 Cleaning Methods and Materials The cleaning method chosen should be appropriate for the solder paste flux formulation and state at the time of cleaning (misprinted solder paste or after soldering). For water-soluble solder pastes, removal of the flux residues is critical to the reliability of the device. These solder paste residues typically contain ionic and corrosive constituents. Incomplete removal will lead to damage of the electronic assemblies in the field. Ideally, the deionized water used in the PCB cleaning operation, should be hot. The volume of water and any additional cleaning agents used, should insure complete removal of the flux residue.

There is great interest in removing the residues of low residue solder pastes to widen the process window for future steps, such as conformal coating and potting applications, or some are cleaned just for appearance sake. Most low-residue solder pastes leave a residue that is insoluble in water. Therefore if water alone is used to clean the assembly, many times all that occurs is the flux residue is replaced by a white, powdery residue. In some cases incomplete washing or removal of a low-residue flux can be more detrimental to the PCB than leaving the residue in place. While a wide range of cleaning formulations can be

used for this purpose, aqueous based, pH neutral or near neutral, VOC compliant and spray-in-air formulations are generally preferred for these applications.

However, not all cleaning materials can remove the residues from all pastes or cored solder wire solders. The suppliers of the soldering or cleaning materials very often have data indicating which combinations are most effective. The compatibility of the cleaning agent with the materials of construction is key to successful cleaning. The compatibility should be checked before the process is implemented on the production floor.

11 COMPATIBILITY OF SOLDER PASTE FLUX RESIDUES (PROBE TESTABILITY)

This section provides guidance on evaluating the automatic test equipment (ATE) and in-circuit testing (ICT) compatibility of no-clean solder paste flux residues. ATE uses spring-loaded probes (“pins”), with typical full-compression forces of 2 to 110 grams, which form an electrical contact between the assembly under test, and the equipment performing the test. Solder paste residue may form an insulating barrier, to give false fails during ATE functional testing of assemblies.

Many factors have combined to produce the need for better pin-testable solder pastes. When the majority of surface mount assemblies were cleaned, there was little concern about pin testing. As no-clean processes became increasingly accepted by the industry, accommodations were made for the presence of solder paste residue. Initially, engineers would try to limit pin testing to the “wave” side of the board or, when two-sided reflow became the standard, they would design test lands away from the connections.

However, with package density increasing, particularly for portable personal electronics and telecommunications, the board area “real estate” is now becoming too precious a resource for “off site” test lands. The electronics assembly industry is now faced with cleaning, or finding a true pin-testable paste.

11.1 Test Plan Overview Figure 11-1 and Table 11-1 have been designed to assist users in assessing the needs and the type of tests most appropriate to their assembly process flow.

11.2 Approaches for Improved ATE-Compatibility A flux residue is considered pin-testable if it does not interfere with the test probe’s ability to make electrical contact with its target lead or land, both initially, and after repeated probing (more than 560 times) of the same spot on different boards by the same probe.

The flux formulation approaches taken to reach this ideal goal are multifarious, and each has its advantages and drawbacks.

Zero Residue Paste Even with inert atmosphere reflow ovens, and/or forming gases, the technology does not currently exist for zero residue paste, although attempts have been made to develop these, either using materials that volatilize at soldering temperatures, or polymers that break down, to form volatile fragments during reflow.

Ultra Low Residues (ULR) Most ULR pastes (that is, those that leave a residue of less than 4.0% by weight of the paste as residue) mostly have very hard impenetrable residues, which can cause difficulties with ATE, particularly when testing on through-hole vias. They may also have other processing difficulties, such as a short stencil life or tack life.

Off-Fllet Residues or No-Clean Tack Fluxes In these formulations, a surface active material has been added to make the residues flow away from the soldered connection. These types of residue may be acceptable when tests are done only on component terminations, but this approach does not work for filled, particularly deeply concave, vias.

Friable Residues Flux residues that shatter easily may be found to work in the short term with single point probes, but there is a potential here for flux build up, especially on crown-point probes, where the end of the probe can entrap and accumulate fragments of the residue. The residues of these pastes are often considered cosmetically unacceptable after testing.

Soft Residue Pastes The majority of pastes designated as pin-testable are of this design, where a low molecular weight, low crystalline oligomer (short polymer) or other softening material has been added to the flux, to increase the penetrability of the residue. Some solder pastes have fluxes that are entirely based on the use of polymers. Synthetic polymers with polar end groups are potential candidates in this category. One of the major disadvantages of these more polar materials is that they may hydrogen bond to water (in humid conditions) thus creating a soft residue that may pave the way for other contamination issues.

11.3 Main Characteristics Investigating the following areas is crucial in ensuring the paste is compatible with the envisioned process:

- ***Penetrability*** The flux residue must allow the test probe to make electrical contact with the lead or land.
- ***Repeatability/Build up Potential*** The flux residue must not interfere with subsequent testing by the same probe, or the repeated testing of the same board.

11.4 Major Control Variables Most importantly, the test board should be exposed to either the same processing conditions as the product experiences, or “worst case” conditions, if the paste is to be used for a variety of applications. The determination of what constitutes “worst case” depends on the paste type.

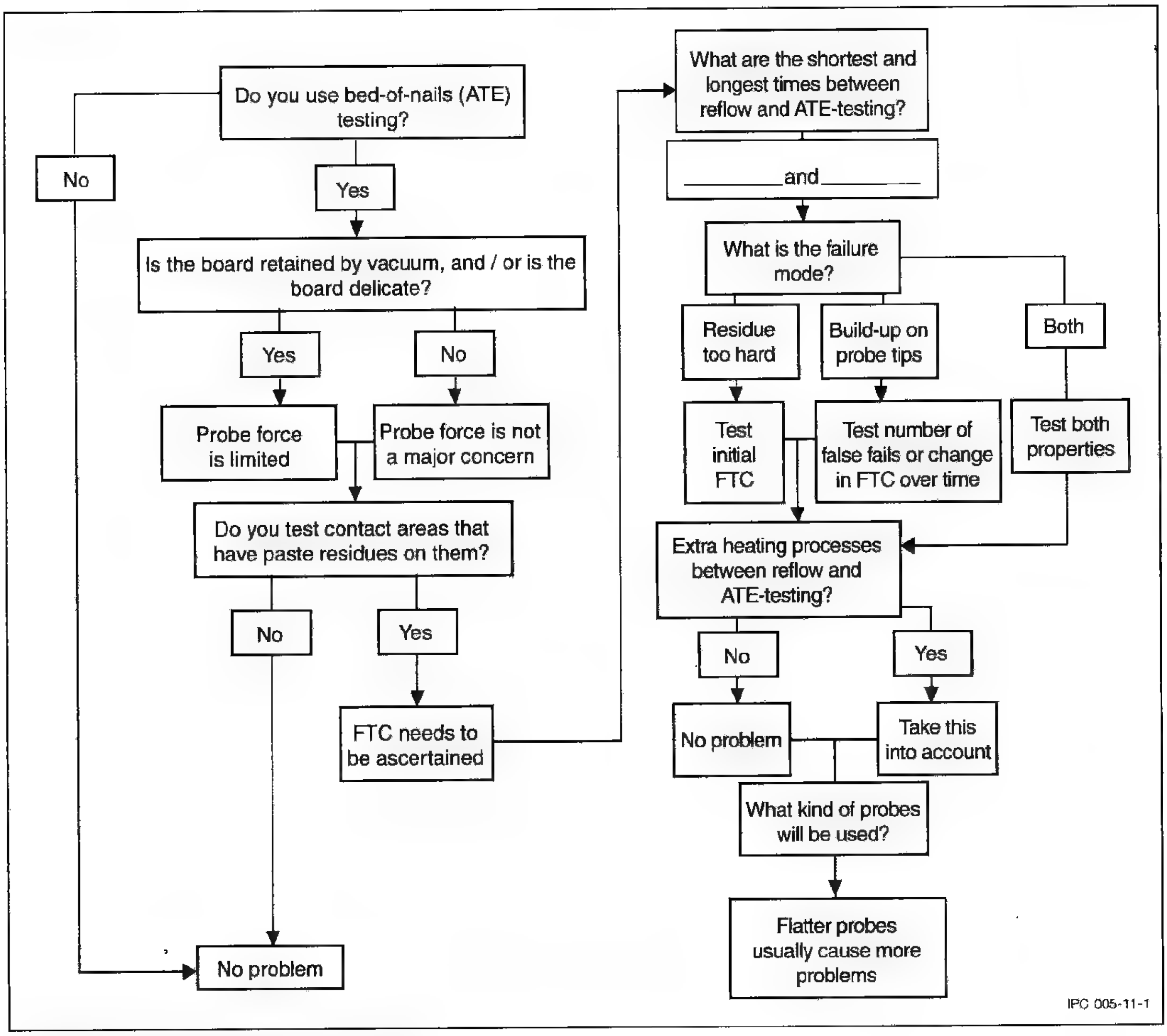

Figure 11-1 Overview for ATE Testing Plan

Table 11-1 Questions for Probe Testing Flow Chart/Guide

1. Do you use bed of nails testing?	[ ] Yes	[ ] No
2. What type is it?	[ ] Vacuum?	[ ] Fixtured (clam shell)?
3. Do you test on dedicated test points?	[ ] Yes	[ ] No
4. Do you test points that may have paste residues on them (including pins or pin through paste)?	[ ] Yes	[ ] No
5. What is the shortest time between Reflow and testing?	[ ] <3 hours [ ] <8 hours	[ ] <24 hours [ ] More
6. What is the Maximum time between Reflow and testing?	[ ] <3 hours [ ] <8 hours	[ ] <24 hours [ ] <72 hours [ ] More
7. What is the failure mode?	[ ] Failure to contact caused by residue hardness	[ ] Residue build-up on probe [ ] Combination
8. Are there additional processes between reflow and testing?	[ ] Hand soldering [ ] Wave soldering	[ ] Double-sided reflow [ ] Other heating process(es) _____
9. What are the probe types used?	[ ] Point [ ] Chisel	[ ] Crown [ ] Twisting crown

11.5 Residue Open Time The engineer needs to determine the minimum, average, and maximum time-delay between paste reflow and pin testing. Many soft residue pastes are designed to harden over time, to reduce the tackiness of the residue and so lower the potential for attracting contaminants. If a formulation hardens in 3 days, and the application calls for up to seven days between reflow and probing. The probes test should be tested initially and over time to identify probe surface effects, if present.

Finally, it should be noted that some residues, such as those of the high residue polymer based pastes, may also be found to be sensitive to the humidity of the environment in which they are stored, softening in a more humid environment.

11.6 Temperature The effect of temperature must not be ignored. Warm residues are usually softer. The reflowed assembly must also be allowed time to cool to room temperature to test the residue.

11.7 Reflow Conditions When the assembly is subjected to double-sided reflow and/or a wave process, the properties of the residue are different than if it were only subjected to one heat cycle. If the solder paste has been designed to give "soft" residues, then a higher spike temperature may make the residue harder. Other reflow conditions, such as the difference between forced convection and standard infrared (IR) reflow can also affect the rheological characteristics of the residue. The increased air-velocity and increased ramp rates seen with forced convection may lead to harder, less penetrable residues. The use of inert atmospheres for reflow has been found to give softer, more penetrable, residues.

11.8 Test Points If the majority of test points are filled vias, the penetrability and build up potential issues are of greater concern. When designing the test, the engineer must consider this along with the following items:

- Test Probe Design and ATE Equipment
- Probe Build-up
- Residue Dryness

11.8.1 Test Probe Design and ATE Equipment Most boards have a wide variety of test points that require the use of different probe types. The engineer must make sure that representative samples of probe types are used in the evaluation phase to ensure compatibility. The choice of ATE equipment with a "clam shell" or vacuum fixturing also influences the maximum amount of probe-force that can be used to penetrate hard flux-residues on an assembly.

11.8.2 Probe Build-up If the ATE equipment is in-line, and run on 100% of the boards, a test for residue build-up should be performed. An acceptable standard should be 90% successful hits over a 560 hit series or an agreed successful hit rate as agreed by customer and vendor.

11.8.3 Residue Dryness The IPC test for the ability of flux residue to pick up particulate matter is IPC-TM-650 - Method 2.4.47: Flux Residue Dryness. The end use conditions of a product may dictate that this test be performed. If an assembly is in an air draft or used in dusty conditions, the residue may attract and retain airborne particulate contaminants, which is a cause of concern for SIR and/or electromigration.

12 SHELF LIFE

This section lists a series of tests to be performed for determining the shelf life of a solder paste material; see Table 12-1.

12.1 Test Procedures This test is relevant for the solder paste manufacturers only to establish the shelf life of the material.

- ***Specimen*** Prepare material to be tested in sufficient quantity for conducting the above tests in multiples; at least 10-12 jars of 500-700 g/jar each are required, one jar per scheduled tests above and discard material after the above tests are completed. The material should be stored in the condition (e.g., 0 - 7°C [32 - 45°F] for refrigeration or 25°C ± 1°C [77.0°F ± 1.8°F] for ambient temperature) intended for stability or shelf life of the material.
- ***Preparation*** Before testing, the solder paste sample shall be stabilized to 25°C ± 1°C [77.0°F ± 1.8°F]. At 25°C ± 1°C [77.0°F ± 1.8°F], using a spatula, stir by hand for 30 seconds (about 30 turns), until uniform consistency is obtained. Re-adjust sample to 25°C ± 1°C [77°F ± 1.8°F].
- ***Test*** Immediately test the material after sample preparation. It is recommended that the tests be conducted in the order as they appeared in Section 2. Print test should always be performed last at that interval, as most of the material will be consumed.
- ***Test Frequency*** Test within 24 to 48 hours after the material is prepared for the initial data and at intervals (biweekly, monthly) thereafter with fresh material of the same batch. Conduct the test up to the intended shelf life of the material. Suggested test intervals of initial, 1st, 2nd, 4th, 6th and 8th weeks, 3rd, 4th, 5th, 6th, 9th and 12th month or other time intervals up to the time span of the intended shelf life.

12.2 Evaluation All test results at each interval should be in accordance to the specific test mentioned above. If any of the test results at that interval fall outside of the desired specification limits, the shelf life of the material is defined as the total time from date of manufacture to the time the material was most previously tested (with acceptable results) at the conditions that this material is stored.

Table 12-1 Test Recommendations for Shelf Life Determination

Test type	Requirement
Physical appearance	The paste should be smooth, homogeneous, no lumps, no crusting and no flux separation.
Solder balling	The solder balling property of the material is determined in accordance with IPC-TM-650, Test Method 2.4.43.
Reflow on copper substrate	The reflow property of material on copper substrate is determined in accordance with IPC-TM-650, Test Method 2.4.45.
Fineness of Grind	Use Fineness of Grind (FOG) Gauge (per IPC-TM-2.2.14.3 or ASTM D 1210) to determine the smoothness of the material. The FOG should be less than the largest size of powder used.
Tackiness	The tackiness properties of material are determined in accordance with IPC-TM-650, Test Method 2.4.44.
Slump test	The slump property of material is determined in accordance with IPC-TM-650, Test Method 2.4.35.
Viscosity (select one) i) Brookfield:	The viscosity property of the material is determined in accordance with IPC-TM-650, Method 2 4.34 & 2.4.34.1.
ii) Malcom :	The viscosity property of the material is determined in accordance with IPC-TM-650, Method 2.4.34.2 & 2.4.34.3.
Print Test	This test method has not yet been developed.
Other	As required

12.3 Control Variables A number of factors can contribute to differences in the shelf-life of a solder paste:

- ***Temperature*** Low temperatures slow the rate of chemical reactions, and also the rate at which solder powder settles in the flux (paste/flux separation). However, if the temperature is too low, some flux components may start to crystallize in the flux; a process that is essentially irreversible.
- ***Humidity*** The presence or absence of trace amounts of moisture can radically alter the way solder paste performs. Typically, water-wash pastes need to be stored in slightly humid conditions, while no-clean pastes are best stored under the driest conditions available. Even "sealed" jars or cartridges of paste may either have permeable seals, or use plastics that are not effective as humidity-barriers over long periods.
- ***Vibration*** Because solder pastes are usually highly thixotropic, excessive vibration or jarring during transportation or storage lowers the viscosity of the flux. This can then lead to a shortened shelf life for the paste, due to increased separation or more rapid concretion.
- ***Conditions During Manufacture*** Depending on how well the solder paste remains sealed away from the atmosphere during mixing and packing, ambient manufacturing conditions (particularly temperature, humidity and dust levels) may be an important factor determining the shelf-life.

APPENDIX A

Presented at IPC SMEMA Council APEX® 2000

Mass Reflow Assembly of 02\01 Components

James H. Adriance
Universal Instruments Corporation
Binghamton, NY

Jeffrey D. Schake
DEK Screen Printers
Weymouth, England

Abstract

The research summarized in this paper will help to address some of the issues associated with solder paste mass reflow assembly of 02\01 components. Attachment pad design, stencil design, component to component spacing, component orientation, flux type, and solder paste reflow atmosphere were the major variables researched during the project. The two major responses from the experimentation were assembly yield and assembly quality. Assembly yield defects, such as tombstones (open solder joints), solder beads, and component mis-placement, were used to determine the assembly yield. Solder joint shape, solder appearance, and solder volume (unacceptable low, acceptable or unacceptable high) responses were used to determine the quality of the assembly process. The combination of flux type and reflow environment were found to have the largest impact in the number of assembly defects produced. Boards assembled with no-clean solder paste and reflowed in an air atmosphere exhibited the best yields with the highest tolerance for attachment pad dimension variation. Conversely, assembly processes using no-clean solder paste with a Nitrogen reflow atmosphere generated the largest number of assembly defects and was found to be the most sensitive to changes in the attachment pad design.

Introduction

The need to reduce the size and weight of electronic products is continuing as Surface Mount Technology matures further. Size reduction in both active and passive components coupled with improved printed circuit board technology is producing smaller, lighter weight, and higher performing end products. Extensive research and development continues to reduce the size of active packages. Passive components have also reduced in size to enable designers to use less printed circuit board area for performing a given task. The use of 06\03 and 04\02 components have been prevalent for a number of years. These component sizes can be run in high volume applications at very high yields. More recently, 02\01 components have been implemented in particular applications. The 02\01 component is approximately one-quarter the size of a 04\02 component. The smaller size of the 02\01 components could potentially reduce assembly process robustness and yield. This paper presents the results of an ongoing study designed to determine the impact that specific assembly and board design parameters have on the yield of the solder paste mass reflow assembly of 02\01 components.

A full factorial experimental design of 27 different attachment pad designs (3 levels each for distance between pads, pad width, and pad length) were used to determine the optimum attachment pad design. Five different stencil aperture designs were tested for each attachment pad design. No-clean and water-soluble flux chemistries were tested in both air and Nitrogen reflow environments. Component to component spacing was tested at four different levels at both zero and ninety degree component orientation. Stencil thickness, stencil fabrication, attachment pad metallurgy, solder mask type, screen printer process settings, thermal profile, reflow system, and component placement system were major parameters that were fixed during the research project.

Experiment Materials & Assembly Equipment

A test board with both 02\01 and 04\02 components was designed for the experimentation. Figure 1 is a photograph of the 02\01 test vehicle. The printed circuit board was a single sided panel that measured 7.5″ wide by 12.5″ long. The board thickness was a standard 0.062″. Attachment pad metallurgy was bare Copper covered by Entek Plus (OSP). Half ounce Copper was used for all traces and attachment pads. Taiyo PRS4000 was the solder mask used. Three different attachment pad widths, lengths, and spacing between pads were tested for both the 02\01 and 04\02 components in a full factorial design, giving a total of 27 different attachment pad designs for both the 02\01 and 04\02 components. Each pad design was replicated 120 times within a single row. Each row was designated a three letter code based on the attachment pad dimensions from Table 1. An example for a 02\01 attachment pad design would be ADG (pad width A = 0.012″, pad length D = 0.008″, and pad spacing G = 0.009″). Four different component to component spacing 0.008″, 0.012″, 0.016″ and 0.020" were tested. Thirty components of a given attachment pad design were blocked together to test component spacing. All attachment pad traces were run out through the ends of the attachment pads, enabling only component spacing testing to be conducted between the component side to side (not end of component to end of component).

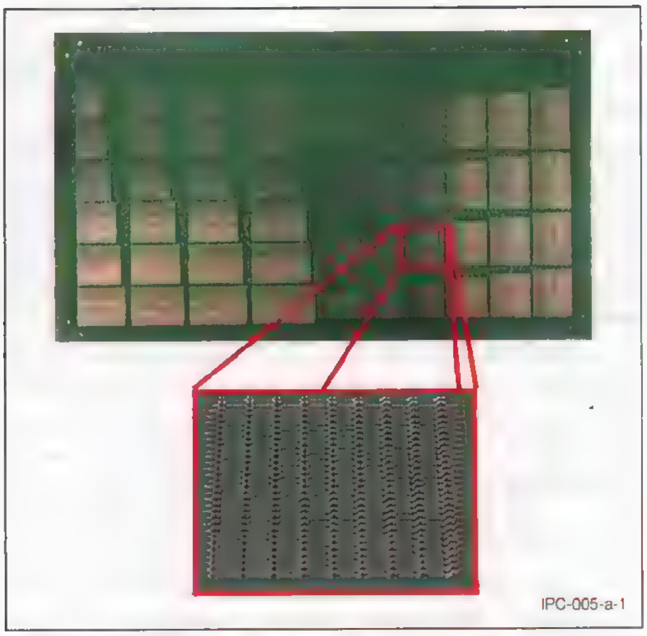

Figure 1 Photograph of 02/01 Test Vehicle

Table 1 Attachment Pad Dimension Matrix for 02\01 Components

02\01 Pads Designs			
Pad Width = W	A = 0. 012″	B = 0.015″	C = 0.018″
Pad Length = L	D = 0.008″	E = 0.012″	F = 0.016″
Spacing between pads = S	G = 0.009″	H = 0.012″	I = 0.015″

The test vehicle was designed at both zero and ninety degree component orientation for all designs. A fully populated test vehicle contained 12,960 components. Table 1 lists the 02\01 attachment pad dimensions for all three levels. Figure 2 shows the dimensioning legend for the 02\01 attachment pads.

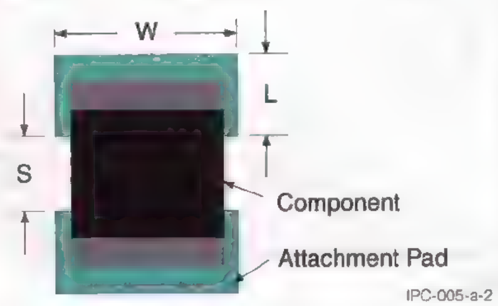

Figure 2 Attachment Pad Dimensioning

All solder paste printing for the experiment was conducted using 0.005″ thick stainless steel laser cut stencils. The stencils were not micro-etched or surface finish plated. A thickness of 0.005″ was selected as a compromise between a 0.004″ thick and 0.006″ thick stencil. The thinner 0.004″ stencil would provide better solder paste release for 02\01 paste deposits, but would inherently reduce the solder paste volume available for other surface mount devices that are typically found on most applications. A 0.006″ thick stencil was not considered because of unacceptable solder paste transfer that would result for 02\01 components. The metal mask was center justified mounted in a 29″ by 29″ stencil frame. Two stencils were manufactured for the project. Stencil 1 was designed for the first experiment (filter). Five different stencil aperture openings were tested for each attachment pad design. Stencil 2 was designed based on the results from stencil 1. Only one stencil aperture size was used for a given attachment pad design for stencil 2. Table 2 contains the stencil aperture size and aperture position for stencil 2. Figure 3 shows the three different types of stencil aperture positions that were used relative to the center of the component.

Table 2 02\01 Stencil Aperture Size and Position for Stencil 2

Attachment Pad Design	Stencil Aperture Size	Stencil Aperture Position
ADG	0.015″ X 0.009″	0.001″ shift outward
ADH	0.015″ X 0.009″	Centered
AEG	0.015″ X 0.011″	0.003″ shift outward
AEH	0.015″ X 0.013″	0.0005″ shift outward
AFG	0.012″ X 0.015″	0.0005″ shift outward
AFH	0.014″ X 0.016″	Centered
BDG	0.018″ X 0.009″	0.0015″ shift outward
BDH	0.018″ X 0.009″	0.0005″ shift outward
BEG	0.015″ X 0.011″	0.0005″ shift outward
BEH	0.015″ X 0.011″	0.0005″ shift inward
BFG	0.015″ X 0.015″	0.0005″ shift outward
BFH	0.017″ X 0.016″	Centered
CDG	0.021″ X 0.009″	0.0015″ shift outward
CDH	0.021″ X 0.009″	0.0005″ shift outward
CEG	0.018″ X 0.011″	0.0005″ shift outward
CEH	0.018″ X 0.011″	0.0005″ shift inward
CFG	0.018″ X 0.015″	0.0005″ shift outward
CFH	0.020" X 0.016"	Centered

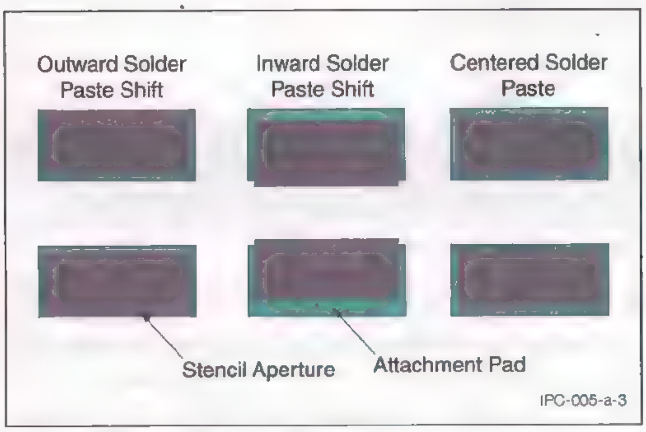

Figure 3 Stencil Aperture Position Relative to Attachment Pad

Both no-clean and water-soluble solder paste formulations were used during the project. Both solder paste types were 90% solids and Type IV powder size. One no-clean and one water-soluble solder paste were selected to provide for the two most common flux chemistry types. Two different solder paste vendors supplied the two solder paste types. The viscosity of the two pastes was approximately 900 KCPS.

A DEK 265 GSX screen printer was used for all solder paste printing. The following screen printer process parameters were used for all stencil printing:

- Print speed = 1.0 inch\sec.
- Squeegee type = metal blades (Transition Automation)
- Squeegee angle = 60 degree
- Squeegee pressure = 2.3 pounds\inch of squeegee
- Print gap = 0 (on contact)
- Separation speed = 0.02 inch\sec.

All component placement for this project was performed on a Universal 4796R HSP. The machine was equipped with the 02\01 option, which includes nozzles, component camera lighting and feeders to handle 02\01 components. All components were fed from tape and reel. Two local fiducials were used for board alignment.

All solder paste reflow was performed in a Heller 1800W forced convection oven. The reflow system contained 8 heating and 1 cooling zone. Oxygen levels within the oven were less than 50ppm. Figure 4 is the thermal profile that was used to reflow all boards assembled during the project.

Assembly defect inspection was conducted by visual inspection under a semi-automatic optical system. All defects were manually recorded and visually verified.

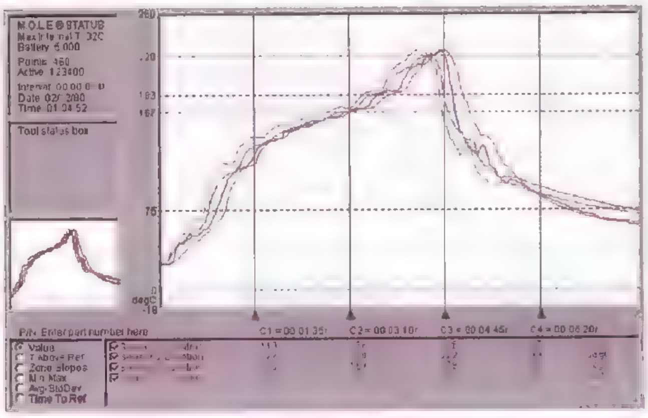

Figure 4 Thermal Reflow Profile

Results

The project was conducted by performing two experiments. The first experiment, which was a filter experiment, was based on running four different processes. The four processes were no-clean and water-soluble solder pastes run in both air and Nitrogen reflow environments. Six fully populated boards were assembled for each of the four processes for a total of 311,040 components. Five different stencil aperture sizes/aperture positions were tested for each attachment pad size.

The second and last experiment was based on running only three of the four processes. The water-soluble solder paste reflowed in Nitrogen was dropped. The combination of water-soluble flux chemistry and Nitrogen environment reflow is typically not used. Only one stencil aperture design was run per attachment pad design. Table 2 contains the stencil aperture designs. The stencil aperture design was selected based on assembly yield and assembly quality for experiment 1. All of the largest spacing between attachment pads (I = 0.015″) were dropped from this experiment. This reduced the total number of attachment pads from 27 down to 18 different designs. Data from experiment 1 showed that the widest spacing (I = 0.015″) produced more open solder joints than attachment pads with smaller spacing. A total of fifty boards were assembled for each of the three processes for a total of 1,116,000 components.

Figure 5 shows the assembly yield from the three different assembly processes. The no-clean solder paste reflowed in air produced the fewest assembly defects for a total of 66. The water-soluble solder paste reflowed in air produced the next lowest number of defects at 1,499. The no-clean solder paste process reflowed in Nitrogen produced the greatest number of assembly defects at 5,665. Figure 5 shows that assembly defects increase when Nitrogen atmosphere reflow is used and when solder paste flux activity is increased (water-soluble solder paste).

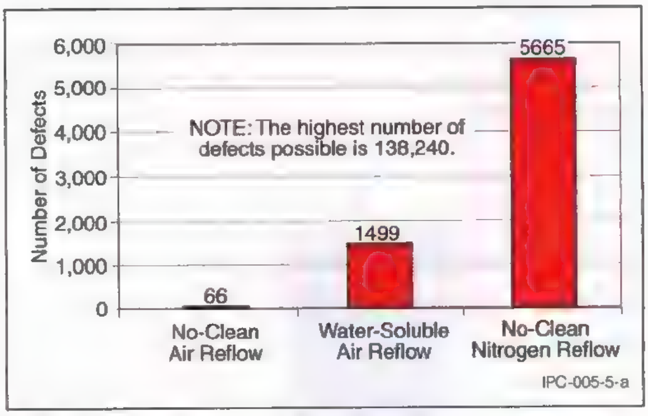

Figure 5 02\01 Assembly Yield Per Assembly Process Type

Figure 6 shows the assembly failure mode distribution for each of the three different assembly processes. Tombstones (open solder joints) and solder bridging were the two main assembly defects. Figure 6 shows that the water-soluble solder paste process reflowed in air produced the lowest percentage of solder bridges at 7.0%, followed by the no-clean solder paste process reflowed in Nitrogen at 15.0%. The no-clean solder paste process reflowed in air produced the largest percentage of solder bridges at 21.0%.

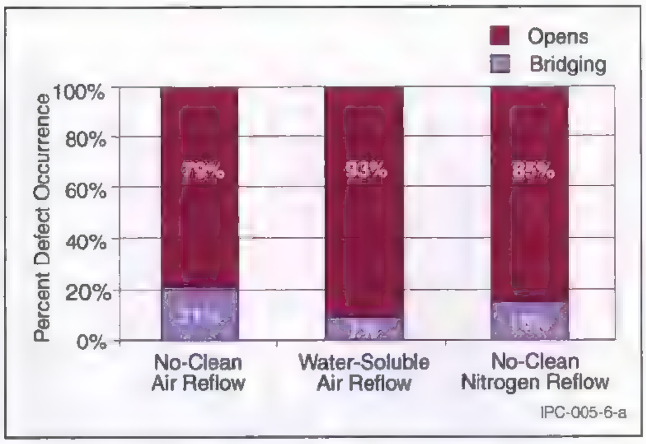

Figure 6 Assembly Failure Mode Distribution by Assembly Process

Figure 7 shows the relationship between solder bridging and component to component spacing for the three different assembly processes. Figure 7 shows that no solder bridging is recorded for any of the assembly processes at a spacing of 0.012″ or larger. Figure 7 also shows that the no-clean solder paste process reflowed in air produced the fewest solder bridges for a total of 14. The water-soluble solder paste process reflowed in air produced the next largest number of solder bridges at 99. The no-clean solder paste process reflowed in Nitrogen produced the greatest number of solder bridges at 866. Twelve attachment pad designs out of 18 did not produce any solder bridges at the smallest spacing of 0.008″ for the no-clean solder paste process reflowed in air. Ten attachment pad designs out of 18 did not produce any solder bridges at the smallest spacing of 0.008″ for the water-soluble solder paste process reflowed in air. Six attachment pad designs out of 18 did not produce any solder bridges at the smallest spacing of 0.008″ for the no-clean solder paste process reflowed in Nitrogen.

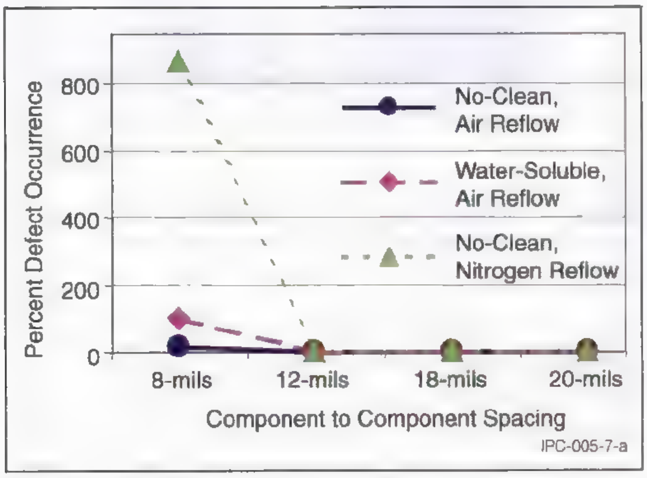

Figure 7 Solder Bridging Defects by Component to Component Spacing and Assembly Process Type

Attachment pad design AEG (L = 0.012″, W = 0.012″, S = 0.009″), which contained the largest distance between solder paste deposits of 0.016″, produced the fewest solder beads. Solder beads are reduced when the distance between solder paste deposits is increased. The amount of solder paste displacement by the component during placement is reduced when the distance between solder paste deposits is increased.

An analysis of paired samples was used to determine if component orientation (0 degree & 90 degree) significantly influenced assembly yield. Zero degree orientation is represented by both component terminations going through the oven at the same time (parallel to the source of heat). Ninety-degree orientation is represented by one component termination going through the oven ahead of the second termination. The hypotheses tested were:

Null hypothesis: $Z = 0$: There is no statistically significant difference in the number of assembly defects between the 0 degree and the 90 degree orientations.

Alternate hypothesis: $Z \neq 0$: There is a statistically significant difference in the number of assembly defects between the 0 degree and the 90 degree orientations.

The 't' test used: [1]$t = (\sqrt{n} \times u)/s$

The p-value for the no-clean solder paste process reflowed in air was 0.5765. Given the high value of p, we fail to reject the null hypothesis. Hence the no-clean solder paste process reflowed in air showed no significant difference in assembly yield when considering component orientation. The lower flux activity of the no-clean solder paste when reflowed in air does not increase the risk of tombstones (open solder joints). The p-value for the water-soluble solder paste process reflowed in air was 0.001959. Given the low value of p, the null hypothesis was rejected. The increased flux activity in the water-soluble solder paste when compared to the no-clean solder paste produced a significant increase in tombstones (open solder joints) for the components that were oriented at ninety-degrees. The p-value for the no-clean solder paste process that was reflowed in Nitrogen was 0.000002. Given the very low value of p, the null hypothesis was again rejected. The use of Nitrogen increased the number of tombstones in the ninety-degree orientation. The vast majority of open solder joints were on the component termination that was reflowed second (trailing termination). The use of Nitrogen increased the surface tension of the molten solder and thus produced open solder joints at a significantly higher rate for components orientated at ninety-degrees versa zero degrees.

Figure 8 shows the assembly defects by attachment pad design for the no-clean solder paste process reflowed in air. Seven attachment pad designs (BDH, BEG, BFG, BFH, CDH, CEH & CFH) out of the 18 did not produce any assembly defects. Based on degree of difficulty for solder

paste printing, solder joint shape, and attachment pad size, designs BEG and CEH are preferred. The smallest attachment pad designs require a smaller stencil aperture design that will tend to clog faster than a larger stencil aperture. Stencils designed at 0.004″ in thickness will reduce 02\01 stencil clogging, but other surface mount devices that require more solder may result in marginal or insufficient solder volume. Solder joint fillet shape on the smallest attachment pad designs did not produce the desired concave solder fillet shape. The largest attachment pad designs are good for solder paste release from the stencil aperture and also produce acceptable solder joint fillet shapes, but the larger attachment pad designs require more printed circuit board space.

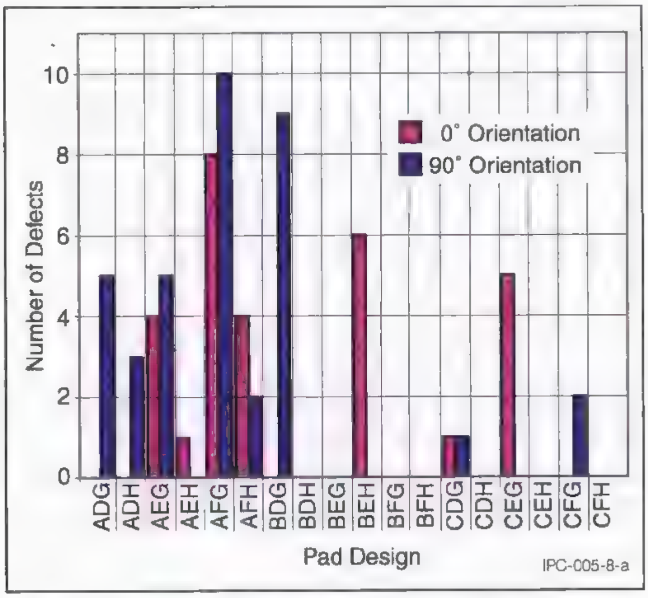

Figure 8 Assembly Defects by Attachment Pad Design for the No-Clean Solder Paste Process Reflowed in Air

Figure 9 shows the assembly defects by attachment pad design for the water-soluble solder paste process reflowed in air. The water-soluble solder paste process reflowed in air produced defects on all attachment pad combinations when considering both component orientations. Attachment pad CEG produced the fewest assembly defects. Attachment pad design CDH did not produce any defects in the zero degree orientation but did produce a relative high number of assembly defects in the ninety-degree orientation. Attachment pad design CEG produced good solder joint shapes and does not occupy as much printed circuit board space as the larger attachment pad designs. Solder paste clogging in the stencil aperture does not represent a problem for this attachment pad design.

Figure 10 shows the assembly defects by attachment pad design for the no-clean solder paste process reflowed in Nitrogen. The no-clean solder paste process reflowed in Nitrogen produced defects on all attachment pad combinations when considering both component orientations.

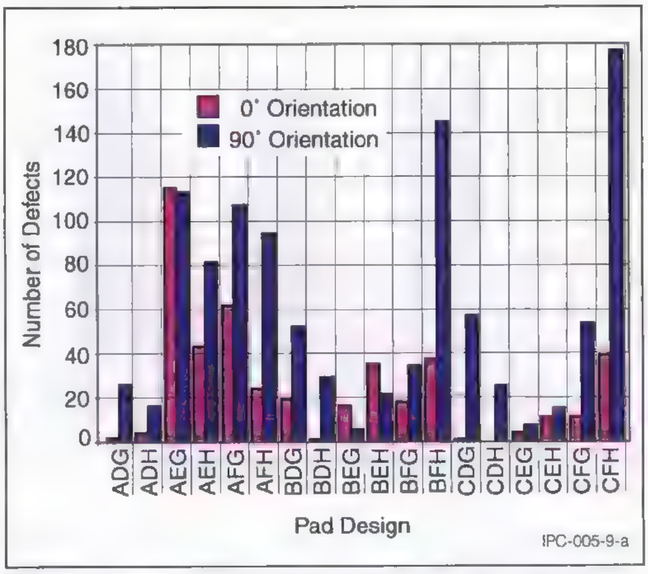

Figure 9 Assembly Defects by Attachment Pad Design for the Water-Soluble Solder Paste Process Reflowed in Air

Attachment pad CEG produced the fewest assembly defects. Attachment pad design CEG also exhibits good solder joint shape and does not occupy as much printed circuit board space as the larger attachment pad designs. Solder paste clogging in the stencil aperture does not represent a problem for this attachment pad design.

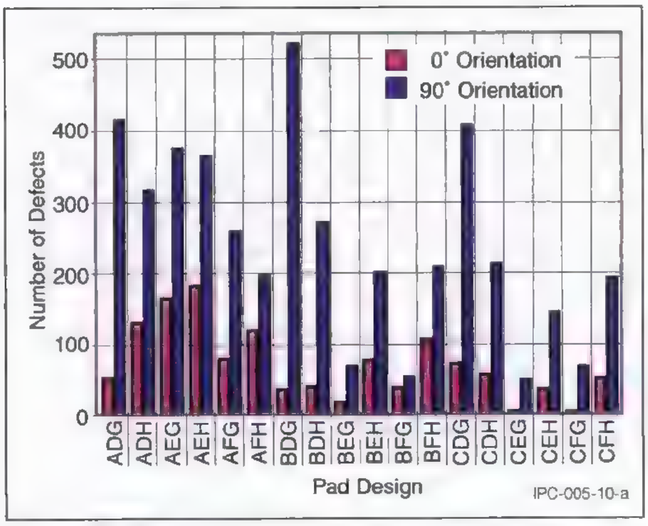

Figure 10 Assembly Defects by Attachment Pad Design for the No-Clean Solder Paste Process Reflowed in Nitrogen

Figure 11 shows the number of solder joint defects tracked by attachment pad width and assembly process type. This data was generated based on the optimal attachment pad designs with respect to each assembly process type, holding the corresponding attachment pad length and attachment pad spacing parameters constant, and varying the attachment pad width across all experimental levels. Generally for the three assembly process types, the tendency is

for yield to improve as the attachment pad width increases. Similarly, among all assembly process types the defect levels are more sensitive to attachment pad widths between 0.012″ and 0.015″. For both the water-soluble process reflowed in air and the no-clean process reflowed in Nitrogen, the minimum number of solder joint defects are achieved at the highest level 0.018″ attachment pad width. This trend changes slightly for the no-clean process reflowed in air, where the best yield is actually produced at the intermediate level 0.015″ attachment pad width. However, due to the limited number of defects found across the boards built by this assembly process type, the difference in the defect levels between attachment pad widths of 0.015″ and 0.018″ is found not to be statistically significant. Upon identifying the trends according to assembly process type, the yield produced by the no-clean process reflowed in air is least sensitive to attachment pad width, while the no-clean process reflowed in Nitrogen is the assembly process type where yield is most sensitive to attachment pad width variation.

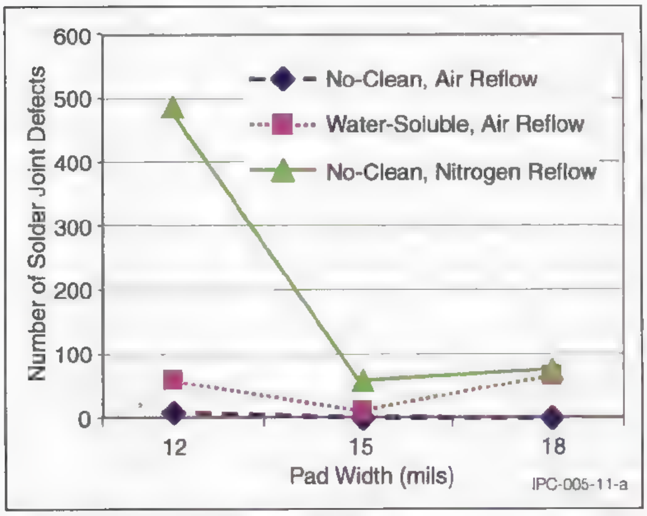

Figure 11 Assembly Defects by Attachment Pad Width and Assembly Process Type

Figure 12 displays the number of solder joint defects that occur as a function of attachment pad length and assembly process type. Similar to the previous graph, this data was generated based on the optimal attachment pad designs with respect to each assembly process type, holding the corresponding attachment pad width and attachment pad spacing parameters constant, and varying the attachment pad length across all experimental levels. The plotted results suggest that the optimal attachment pad length is the intermediate level of 0.012″ for all three assembly process types. Generally, the largest impact on yield is shown to occur between the low and intermediate attachment pad length levels of 0.008″ and 0.012″. The no-clean process reflowed in Nitrogen is clearly the most sensitive process for affecting the number of defects, with a much more substantial dependence on attachment pad length than any other assembly process type. No defects were observed on any of the boards assembled with a no-clean process and air reflow for both the intermediate and high level attachment pad lengths of 0.012″ and 0.016″.

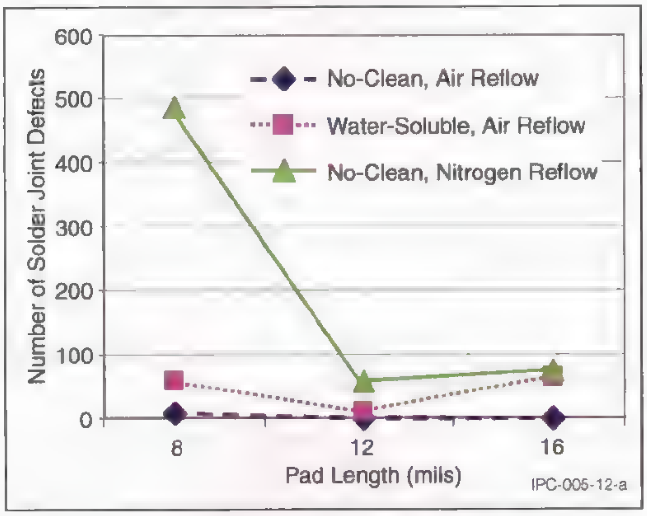

Figure 12 Assembly Defects by Attachment Pad Length and Assembly Process Type

Figure 13 shows the relationship between solder joint defects, attachment pad spacing, and assembly process type. This data was also generated based on the optimal attachment pad designs with respect to each assembly process type, holding the corresponding attachment pad width and attachment pad length parameters constant, and varying the attachment pad spacing across both experimental levels. The three assembly process types all give similar defect trends, with more solder joint failures occurring at the larger level 0.012″ attachment pad spacing. The combination of no-clean process and Nitrogen reflow environment is the assembly process type that is most susceptible of impacting yield based on changes in the attachment pad spacing. The no-clean process reflowed in air is the assembly process most resistant to generating failures that can be attributed to changes in attachment pad spacing.

Conclusions

Of the three assembly processes tested, the no-clean solder paste process reflowed in air produced the fewest number of assembly defects for both tombstones (open solder joints) and solder bridges. The no-clean solder paste process reflowed in air also produced the most attachment pad designs that were free from assembly defects. Furthermore, this assembly process type was found to be the least sensitive (of the three considered in this study) for influencing the number of solder joint defects across a variety of pad designs. The water-soluble solder paste process reflowed in air produced the next fewest number of assembly defects followed by the no-clean solder paste process that was reflowed in Nitrogen. The use of low oxygen levels (under 50ppm) and more active solder paste flux chemistry decreases assembly yield and assembly robustness. Longer

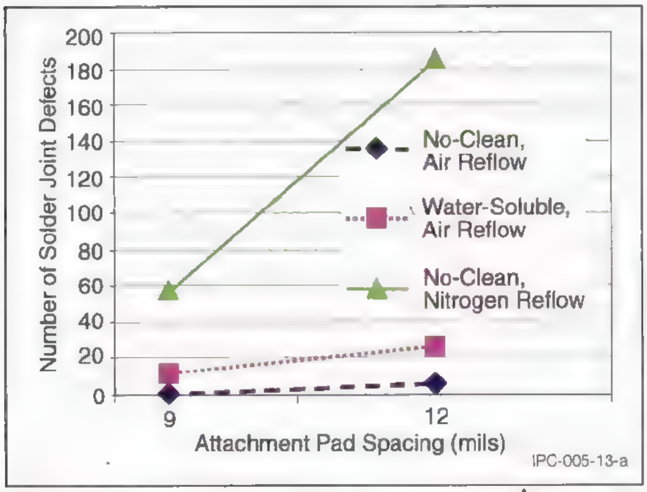

Figure 13 Assembly Defects by Attachment Pad Spacing and Assembly Process Type

thermal reflow profiles may reduce the number of assembly defects for the water-soluble solder paste reflowed in air and for the no-clean solder paste process reflowed in Nitrogen. Higher oxygen content during reflow for the Nitrogen reflow process would most likely also reduce assembly defects. The use of Nitrogen increases solder wetting forces and reduces wetting times.

Component side to side spacing of 0.008″ was achievable for all three processes without producing solder bridges. The use of Nitrogen during reflow and water-soluble solder paste increases the number of solder bridges. Small attachment pad sizes also tend to solder bridge more readily than larger attachment pad sizes. Combinations of either the smallest attachment pad width or smallest attachment pad length increase the probability of solder bridging. Future research will test component to component spacing under 0.008″ to determine the absolute minimum spacing between components.

Solder beads can be reduced or eliminated by reducing the amount of solder paste that is printed under the component terminations. It should be noted that the number of tombstones (open solder joints) increases as the distance between solder paste deposits increases. When designing the stencil, the distance between stencil apertures should be held to a maximum of 0.010″ to 0.012″. "Home plate" or "v-notch" stencil designs were not tested because of the small attachment pad sizes for 02\01 components.

Component orientation was determined to be insignificant for the no-clean solder paste process that is reflowed in air. Component orientation was statistically significant for the water-soluble solder paste process reflowed in air as well as for the no-clean solder paste process reflowed in Nitrogen. Increased flux activity of water-soluble solder pastes, compared to no-clean solder paste and\or reduced oxygen content during reflow, increases the wetting force of molten solder. Components oriented at ninety degrees (one termination reaching the reflow zone before the other) are more likely to tombstone when higher wetting forces and reduced wetting times are experienced.

Seven attachment pad designs out of the 18 tested for the no-clean solder paste process reflowed in air produced no assembly defects. Attachment pad design BEG was selected as the top choice based on attachment pad size, solder joint quality, and ease of solder paste printing. The BEG design also uses the smallest distance between attachment pads. The wider distance between attachment pads for design CEH was the reason this design ranked second. The preferred attachment pad designs from the other two processes also contained the smaller distance between attachment pads of 0.009″. The no-clean solder paste process reflowed in air is a more robust process when compared to the other two processes. Fewer numbers of acceptable pad designs are available for the other two processes. Attachment pad design CEG produced the best assembly yield for both the water-soluble solder paste process reflowed in air and the no-clean solder paste process reflowed in Nitrogen. The only difference in the design of BEG and CEG is the pad width difference of 0.003″. Increasing attachment pad width and decreasing the distance between attachment pads reduces the amount of component placement accuracy needed and increases the robustness of the placement process. Attachment pad design BEG ranked third for assembly yield for both the water-soluble solder paste process reflowed in air and the no-clean solder paste process reflowed in Nitrogen. Unacceptable assembly yield results were produced from the no-clean solder paste process reflowed in Nitrogen for all attachment pad designs. Unacceptable assembly yield results were also produced from the water-soluble solder paste process reflowed in air for all attachment pad designs when both component orientations are considered.

Research is planned to further investigate assembly placement accuracy and reflow parameter optimization.

References

1. Montgomery, D.C., *Design and Analysis of Experiments - 4th Edition,* John Wiley & Sons, Inc., New York, NY, 1997, p. 51.

Acknowledgements

The authors would like to thank Shravan Jumani a Graduate Research Associate from Binghamton University for support of data analysis, Dii group for support of test board layout and manufacturing, IRI Alphametals for supplying the stencils; Kester and Alphametals for supplying solder paste.

APPENDIX B

Presented at IPC SMEMA Council APEX(r) 2003

Qualification of Solder Beading and Tombstoning in Passive Devices using Designed Experiments

Vijaykumar Ganeshan, Karthik Thenalur and S. Manian Ramkumar, Ph.D.
Center for Electronics Manufacturing & Assembly
Rochester Institute of Technology
Rochester, NY

1 Abstract

Solder beading and tombstoning are observed increasingly with chip components as their size decreases. This is even more crucial in today's packaging, due to the high ratio of passive components in comparison to active components. The increasing number of passive components affects the Defects Per Million Opportunities (DPMO), which inturn affects the overall yield of the assembly line. Hence, it is vital to understand the various causes within the assembly process, which influence the occurrence of these defects. This paper will discuss the results of a process characterization study to understand the effects of solder paste, stencil thickness, board support, reflow profile and the component size on the formation of solder beads and tombstones. A Resolution-V DOE analysis was performed to determine the effect of these factors on the defect occurrence. The response variable for the study was % defects, the ratio of number of defect occurrences to the total number of available defect opportunities.

Introduction

The electronics industry is undergoing significant changes due to the continuous demand for smaller and better products at a lower cost. Product quality becomes a key factor in determining the success of any organization. Product quality is directly related to the process engineer's knowledge of the manufacturing process, the design variables and other parameters affecting the process. Almost every electronic assembly produced today, includes a wide variety of components, among which passive chip components form a major portion of the total number of devices. Hence, any reduction in the number of defects observed in passive chips would be of great savings in PWB assembly.

Some of the common causes for passive component defects are attributed to the formation of solder beads/balls and tombstones. Although some knowledge about the causes for solder beading and tombstoning is available, a properly designed experiment would definitely be beneficial for establishing necessary process control standards, thereby improving the overall product quality and reliability. The experiment described in this paper has been performed with the above-mentioned criteria in consideration, so as to clearly identify the parameters and their settings, that minimize solder beading and tombstoning in 0805 and 0402 chip components. A similar study involving 0201 chip components is also being planned, due to the recent acquisition of 0201 placement equipment.

2 Problem Statement

The purpose of this project is to minimize the occurrence of solder beading and tombstoning in PWB assemblies containing passive chip components. Experimental design techniques are utilized to identify the critical factors/ variables that influence the formation of solder beads and tombstones in surface mount assembly.

3 Solder Beading and Tombstoning

Solder beading is an assembly defect, which occurs when tiny spherical solder balls are attached to a passive component by the flux residue formed during reflow and are usually located at the component base (Figure 1). The solder beads if not removed, may move from their original location (due to impact or vibration) and cause unwanted shorts/bridges between other components, thereby resulting in product failure.

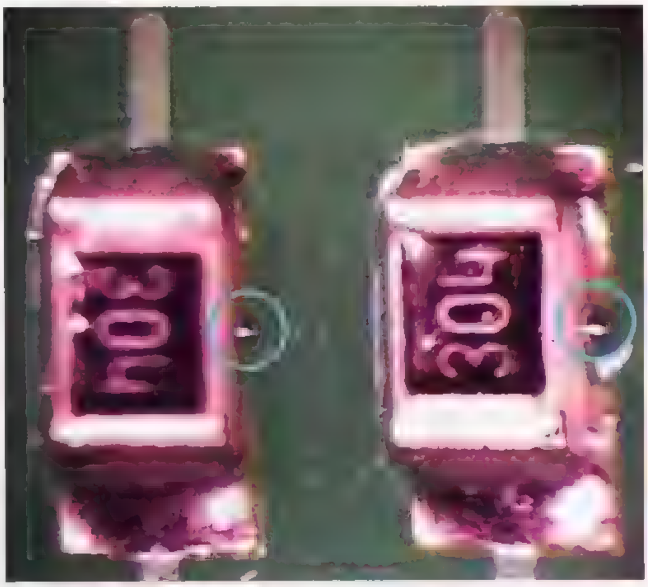

Figure 1 Solder Beading Below a Chip Resistor

Another common defect that is increasingly observed in miniature chip components is tombstoning. Tombstoning is an alignment defect that occurs when a chip flips up from one end of a pad because of unequal wetting forces generated during solder reflow[1,2] (Figure 2). This defect causes

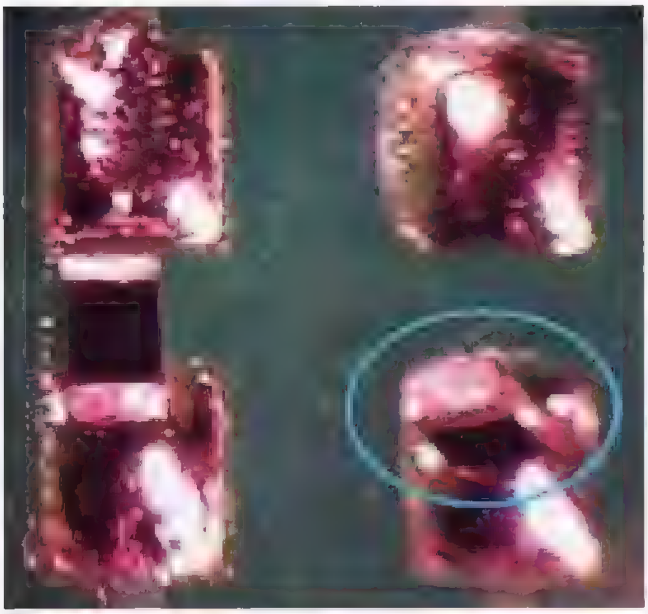

Figure 2 Tombstoning in a Chip Resistor

an open circuit condition and is usually corrected in inspection and rework.

Although both solder beading and tombstoning can be detected in the process of inspection, and suitable rework be performed, the resources for such additional operations in the case of a large-scale manufacturing unit can be costly. The current study aims at identifying the causes for solder beading and tombstoning, and decide on the settings of certain control variables in the surface mount assembly process, that minimize the occurrence of these defects.

4 DOE Formulation

Among the various process issues for passive component assembly, solder beading and tombstoning have been prominent due to their effect on the overall yield of the assembly process. A brainstorming session was conducted to identify some of the common variables that could cause solder beading and tombstoning (Figure 3).

The variables to be considered for the designed experiment have to be set at particular levels in each of the experiment runs, so as to independently determine the effects of each of the factors. This task was achieved by incorporating the relevant factors, in a balanced experimental design matrix using the principles of orthogonality. Table 1 below lists the appropriate factors to be considered for the experiment. The statistical analysis for the DOE was carried out, and is discussed in detail in the following sections.

Table 1 List of Factors Considered for the DOE

Factors		Levels	
Name	Type	-1	1
Stencil-Thickness (Th)	External	4 mil	6 mil
Paste Type (Paste)	External	Paste-1 (Mfr-A)	Paste-2 (Mfr-B)
Reflow Profile (RF)	External	Ramp-Spike	Ramp-Soak-Spike
Board Support (BS)	External	Yes	No

5 Test Vehicle

The experiment was performed using Printed Wiring Boards (test vehicles), which had a HASL finish. The test vehicle consists of various components, that include QFP's, BGA's, CSP's and 0805, 0603, 0402, and 0201 chip components. However for the purpose of this study, only the 0805 and 0402 components were assembled and analyzed for solder beading and tombstoning. The test vehicle (Table-2) consisted of 240 numbers of 0805 and 0402 components each, distributed equally among the four zones: North, South, East, and West.

6 Experiment Execution & Data Collection

The experiment execution involved the following steps.

Solder Paste Printing

The test vehicles were printed with solder paste using a fully automated stencil printer with the following print parameters as mentioned in Table 3.

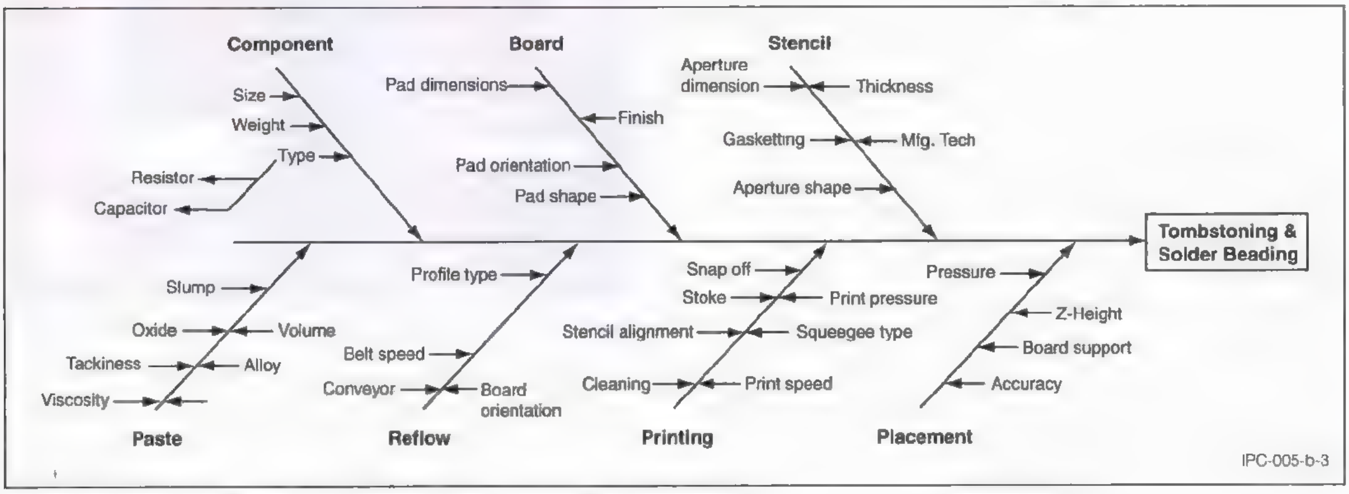

Figure 3 Fishbone Diagram for Solder Beading and Tombstoning

Table 2 Test Vehicle Specifications

Test Vehicle Specifications	Value
Board Material	Standard FR4 - Glass Fiber reinforced laminate
Board Finish	HASL
Board Dimensions	10.5″ X 8.5″ 62-mil Thick
Board References	Global and Local Fiducials

Table 3 Standard Print Parameters Used

Parameter	Setup Value
Print Pressure	12Kg
Print Speed	30 mm/sec
Print Direction	Forward
Separation Speed	3 mm/sec
Snap-Off	0
Squeegee	12″- Metal, 45° attack angle

The settings were kept constant throughout the experiment and were known to provide consistent good quality prints from earlier experiments.

Paste Inspection

The print quality on the test vehicle was inspected visually for any qualitative anomalies like bridging, missing bricks, etc., and the 3-D paste volume was measured using a laser inspection system.

Component Placement

The 0805 and 0402 chip components were assembled using a flexible medium volume pick/place equipment. The placement sequence was optimized for minimum cycle time and multiple nozzles were employed for the same. The components were placed with and without board supports, according to the DOE run order.

Reflow

The test vehicles after component placement were passed through a forced convection reflow oven consisting of multiple temperature zones for reflow soldering. Two different reflow profiles (ramp-spike and ramp-soak-spike profile) were used as part of the experiment, the details of which are provided in a later section. The parameters for the above two profiles were derived from the paste manufacturers specification.

Data Collection

The defects on the test vehicles were identified using an inspection microscope, having a maximum magnification capability of 40X. The defects were identified visually and recorded in data logging sheets for all the test vehicles assembled.

6.1 Full Factorial Data Analysis

A full factorial experiment for the four factors listed in Table 1 requires a minimum of 16 runs to estimate the main effects and the two-way interactions. The design was limited to a single replication since every run combination had 240 numbers of 0805 and 0402 components on the test vehicle. The test vehicles were assembled according to the orthogonal DOE run settings.

Response Variable Used for the DOE

The response variable considered for the analysis was "%Defects." This was calculated using the relation given below:

$$\% \text{ Defects} = \frac{\# \text{ of defect occurences}}{\# \text{ of defect opportunities}} \times 100$$

In this project the total number of defect opportunities available is calculated as shown below:

# of 0805 components per zone	= 60
# of 0402 components per zone	= 60
# of zones per test vehicle (N, S, E, W)	= 4
Total # of defect opportunities per Run per component	= 240

The full factorial analysis is primarily intended to study the influence of the factors on defect occurrence in 0805 & 0402 components individually. The full factorial analysis is carried out by performing the ANOVA using a 95% confidence level for 0805 components as shown in Table 4.

Table 4 ANOVA Results of the Full Factorial Analysis for 0805 Components

Source	DOF	Adj. MS	F	P
Th	1	572.4	4.37	0.091
Paste	1	1330.4	10.16	0.024
RF	1	1100.6	8.41	0.034
BS	1	2.8	0.02	0.889
Th*Paste	1	525.6	4.01	0.101
Th*RF	1	50.1	0.38	0.563
Th*BS	1	312.4	2.39	0.183
Paste* RF	1	56.6	0.43	0.540
Paste*BS	1	73.5	0.56	0.487
RF*BS	1	26.8	0.20	0.670
Error	.5	130.9		
Total	15			

7 Inferences from the ANOVA table for 0805 Components

1. Solder Paste type (Paste) and Reflow profile (RF) have a statistically significant influence on the defect occurrence.
2. Stencil Thickness (Th) and Board support (BS) did not have a statistically significant influence on the defect occurrence.
3. However there is a noticeable change in the %defect levels when these factor levels are switched from one level to another. These inferences are graphically represented in the main effects plot (Figures 4 and 5).

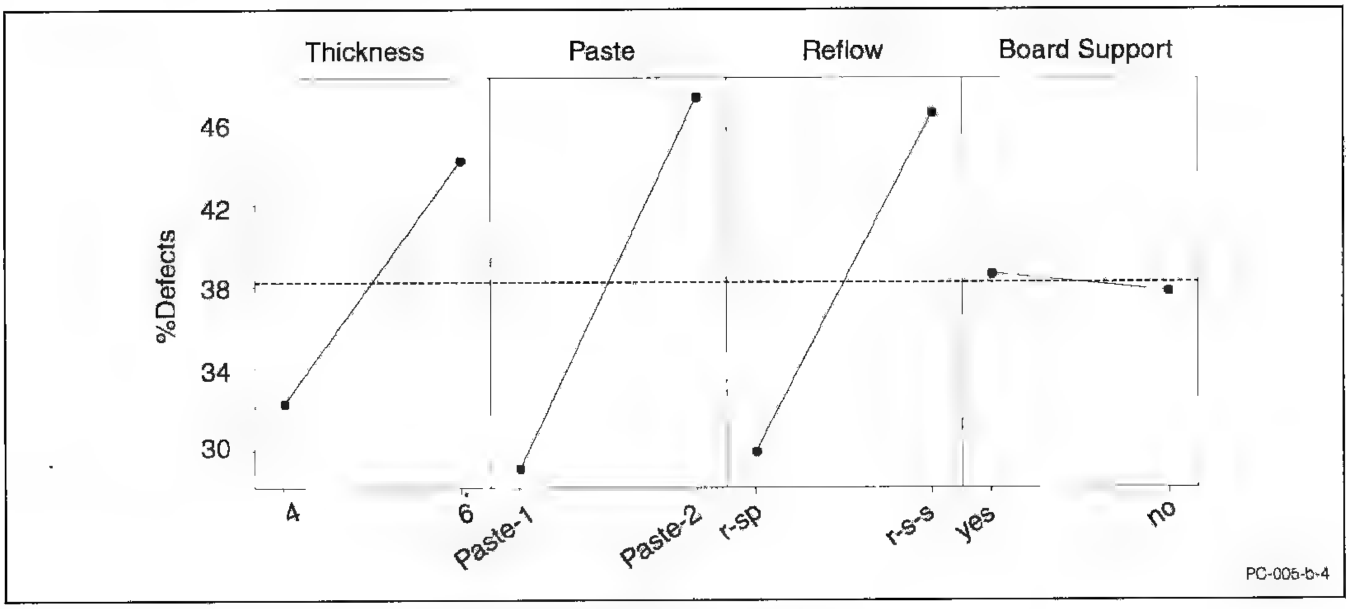

Figure 4 Main Effects Plot of the Full Factorial ANOVA for 0805 Components

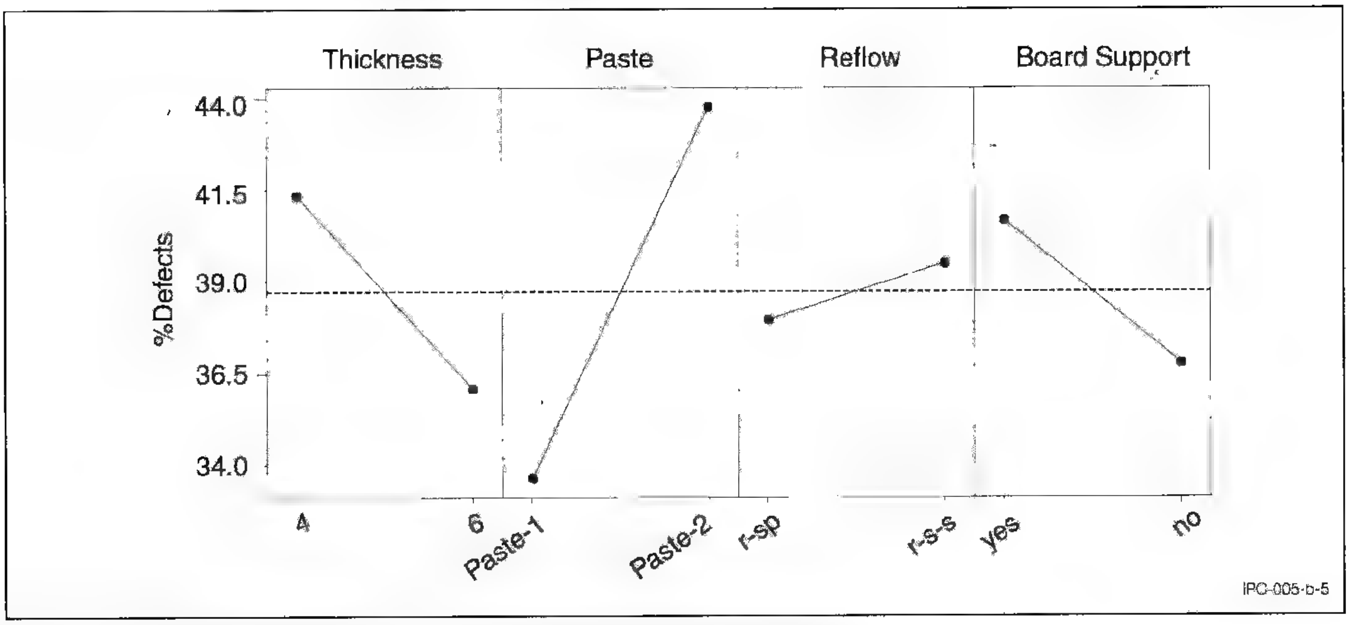

Figure 5 Main Effects Plot of the Full Factorial ANOVA for 0402 Components

The ANOVA table for the full factorial analysis of 0402 components is shown in Table 5.

Table 5 ANOVA Results of the Full Factorial Analysis for 0402 Components

Source	DOF	Adj. MS	F	P
Th	1	111.3	1.09	0.344
Paste	1	408.0	4.00	0.102
RF	1	9.3	0.09	0.775
BS	1	60.1	0.59	0.478
Th*Paste	1	647.7	6.34	0.053
Th*RF	1	1.4	0.01	0.910
Th*BS	1	557.0	5.46	0.067
Paste* RF	1	131.1	1.28	0.309
Paste*BS	1	21.6	0.21	0.665
RF*BS	1	74.0	0.72	0.434
Error	5	102.1		
Total	15			

8 Inferences from the ANOVA Table for 0402 Components

1. Stencil thickness, Solder Paste type, Reflow profile, and Board support do not have a statistically significant influence on %defects observed in 0402 components.
2. However a change in the %defects is observed for a change in each of the factor levels (Figure 5).
3. None of the two factor interactions are statistically significant.

Conclusions and Discussions of the Full Factorial DOE

1. The full factorial analysis performed so far considers only the external factors into the experimental design matrix and analyzes the total defect occurrence in the two component types assembled (0402 & 0805), individually.
2. The results of this analysis indicate that the same set of external factors affect the defect occurrence, in the two component types, at different levels of significance as demonstrated by the largely different significance values in the ANOVA tables (Table 4 and 5).
3. This part of the analysis gives us an insight as to how the two different components react to the external factors in the context of %Defects.
4. The next phase of the analysis, focuses on the types of defects encountered (rather than component type) namely solder beading and tombstoning. To achieve this, the two component types are included into the experimental design matrix as a controlled factor with two levels namely, 0402 and 0805.
5. The inclusion of component size increases the factor count to five and a full factorial DOE for five factors would require a total of 32 run combinations. However a resolution-V design matrix consisting an identical set of 16 run combinations can be used to analyze five factors simultaneously.
6. The defect occurrences were grouped into two groups; beading and tombstoning; and the %defects were calculated as specified in the previous section.

9 Resolution-V Design

The Resolution-V design differs from the original full factorial DOE only in the fifth column (component). The level combinations for the first four factors are the same as the full factorial DOE. The principles of orthogonality says that the addition of a column to a balanced DOE matrix does not affect the orthogonality of the design as long as the fifth factor is set according to the L-16 Resolution-V array. Table 6 lists the set of factors and their respective levels that were used for the experiment.

Table 6 L16 Array for Resolution-V Design

Run #	Stencil Thickness	Paste	Reflow Profile	Board Support	Component
1	-1	-1	-1	-1	1
2	1	-1	-1	-1	-1
3	-1	1	-1	-1	-1
4	1	1	-1	-1	1
5	-1	-1	1	-1	-1
...	....	....	....	....	....
14	1	-1	1	1	-1
15	-1	1	1	1	-1
16	1	1	1	1	1

The Resolution-V design is capable of providing the main effects of five individual factors within a total of 16 runs. However a resolution-V design is not capable of revealing the effects of higher order interactions (3 or more), which are usually not studied when characterizing a PWB assembly process.

10 Resolution-V analysis for solder beading and tombstoning

The resolution-V analysis was performed to identify the factor settings that minimize solder beading and tombstoning in a board containing both 0402 and 0805 components. The %defects were separated based on the defect type and the appropriate values for the corresponding runs were used in the data analysis. Figures 6, 7, 8 & 9 show the main effects plot and the corresponding Pareto charts for solder beading and tombstoning respectively.

11 Inferences from Resolution-V analysis on Solder Beading

1. Paste, reflow profile, and component type are the statistically significant main effects for beading whereas, stencil thickness and board support are not statistically significant as inferred from the pareto chart (Figure 7)

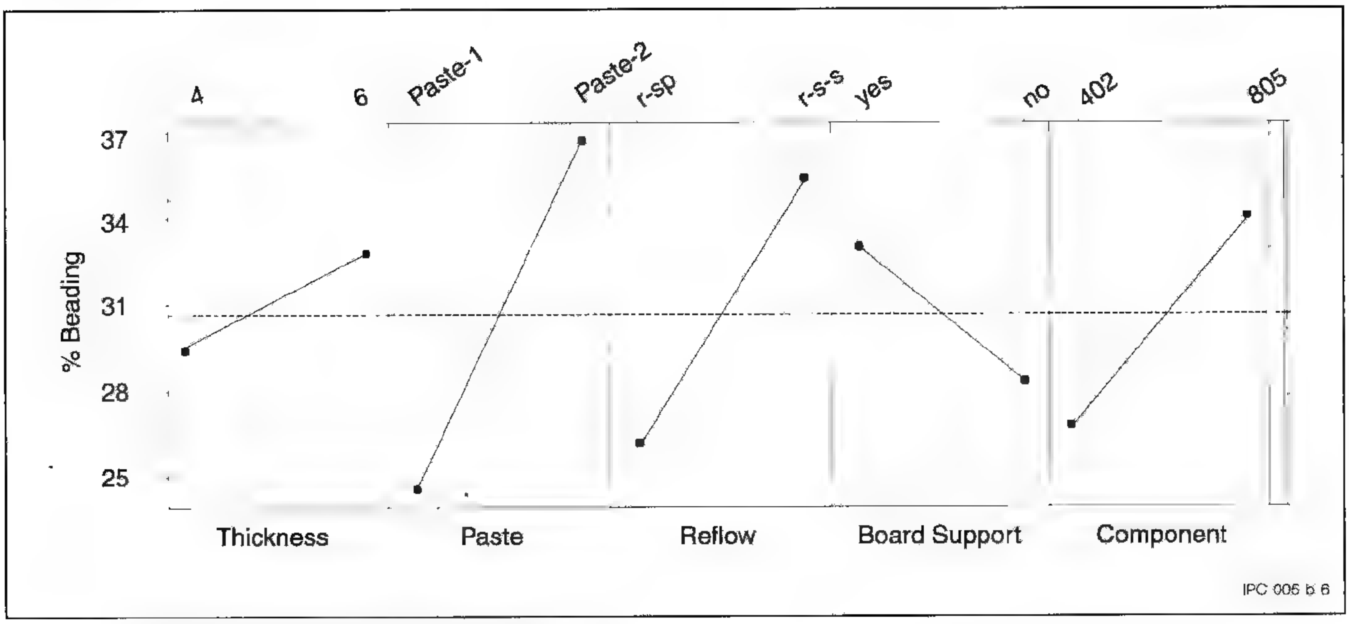

Figure 6 Resolution-V Main Effects Plot for % Beading

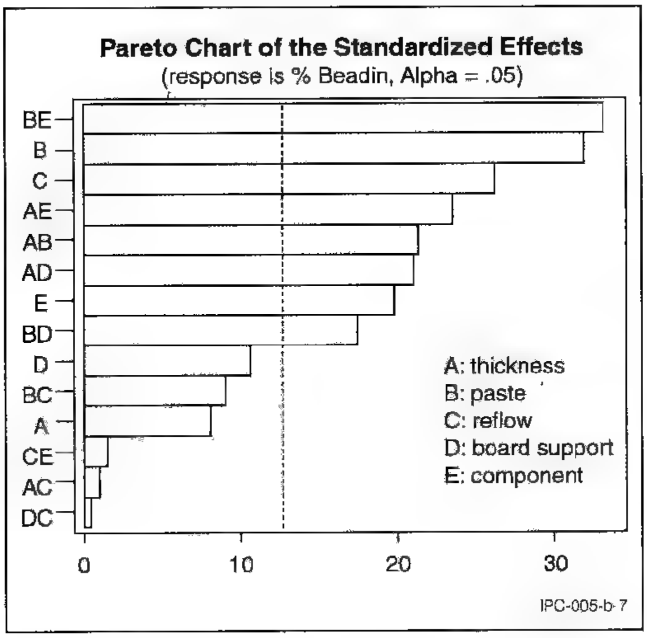

Figure 7 Pareto Chart for % Beading

2. The reasoning as to why a certain level of a particular factor causes reduced beading than the other level is discussed in detail in the following summary section on resolution-V analysis (i.e., for eg. why a ramp-spike profile results in reduced beading than ramp-soak-spike profile).
3. The pareto chart also shows that certain two factor interactions like paste and component type (BE) are significant. These effects will be evaluated extensively as part of a future study, by formulating additional designed experiments with increased number of levels and replications for these factors, enabling us to understand these effects in detail.

12 Inferences from Resolution-V Analysis on Tombstoning

1. Component type was the only significant factor in the case of tombstone formation. Paste, reflow profile, board support and none of the two factor interactions, are statistically significant as inferred from the pareto chart (Figure 9)
2. The reason behind the high significance of component type in the pareto chart (Figure 9) can be comprehended easily by referring to the main effects plot (Figure 8) which shows that the %tombstones for 0805 components was zero and it was 4% for 0402 components. This agrees with the popular notion that tombstoning is more prominent with miniature devices.
3. It can also be inferred that paste-1 has always resulted in reduced beading and tombstoning when compared to paste-2.

13 Conclusions and Inferences on the Resolution-V Analysis

The following inferences can be concluded from the resolution V analysis and existing literature, about each of the factors with reference to the defect types namely beading and tombstoning:

13.1 Effect of Component Size on Solder Beading

1. Solder beading is more pronounced in larger chip (0805) components when compared to smaller (0402) passive devices, the rest of the assembly parameters being identical between the components (Figure 6).
2. Component size governs the amount of paste displaced from the pads (to the solder mask), i.e., bigger components displace more paste than smaller components thereby increasing the chances of forming a solder bead.

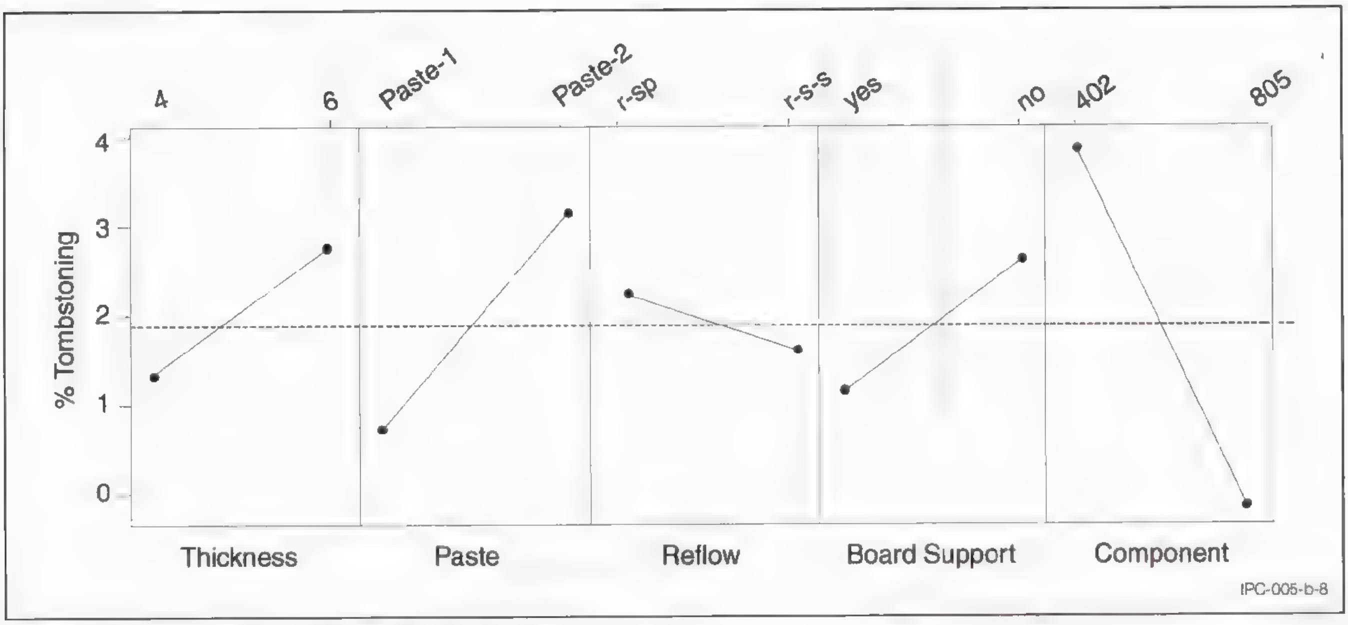

Figure 8 Resolution-V Main Effects Plot for % Tombstoning

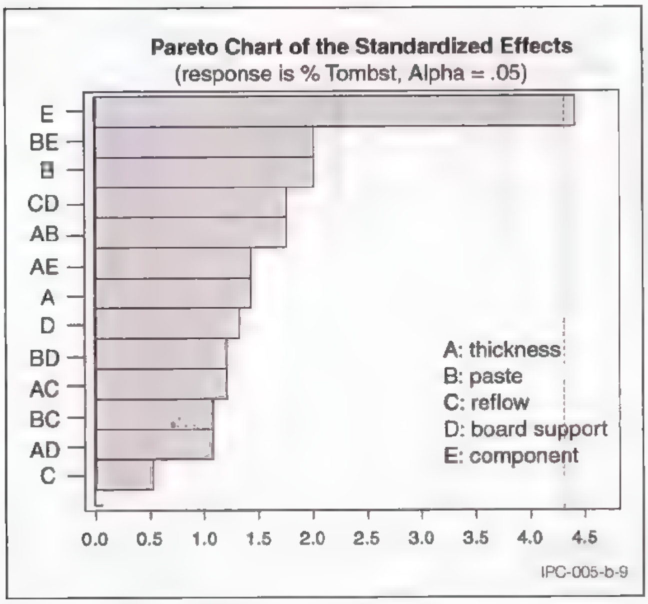

Figure 9 Pareto Chart for % Tombstoning

The displaced solder paste coalesces into discrete beads during reflow.

14 Effect of Stencil Thickness on Solder Beading

1. Thinner stencils (4-mil) result in lesser solder beading than thicker stencils (6-mil) (Figure 6).
2. Excessive solder paste deposited on the component pads increases solder bead formation. Increase in stencil thickness causes a substantial increase in deposited paste volume (Figure 10).

15 Effect of Paste Manufacturer on Solder Beading

1. Paste-1 always exhibited reduced %defects (solder beading & tombstoning) when compared to paste-2.

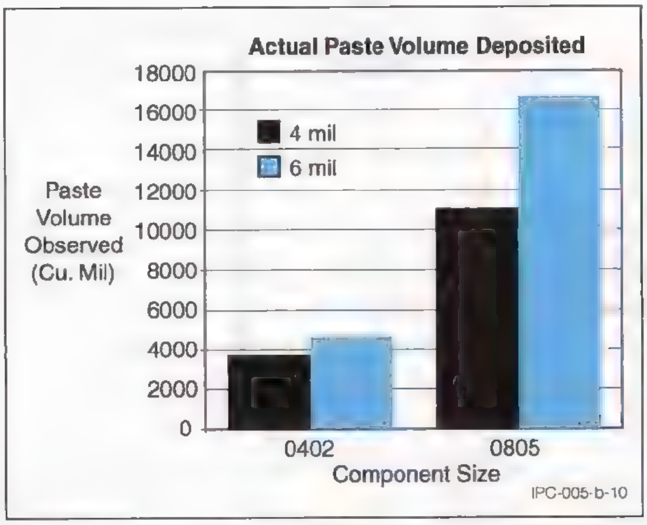

Figure 10 Pareto Chart for % Tombstoning

2. The two pastes used for the study, were from two manufacturers comprising of No-Clean flux and type-III particle size.

16 Effect of Reflow Profile on Solder Beading

1. The ramp-spike (r-s) reflow profile results in reduced solder beading when compared to the ramp-soak-spike (r-s-s) (refer Figures 11 & 12 for profiles).
2. This can be attributed to its gradual ramp up rate when compared to the steep initial ramp rate of the ramp-soak-spike profile, thereby resulting in comparatively less slumping (Table 7).
3. One of the common reasons for solder beading to occur is due to excessive paste slumping on pads during reflow. Ramp up rate affects paste slumping during reflow. A Slower ramp up rate causes an increased loss of volatiles, which increases the paste viscosity.[5]

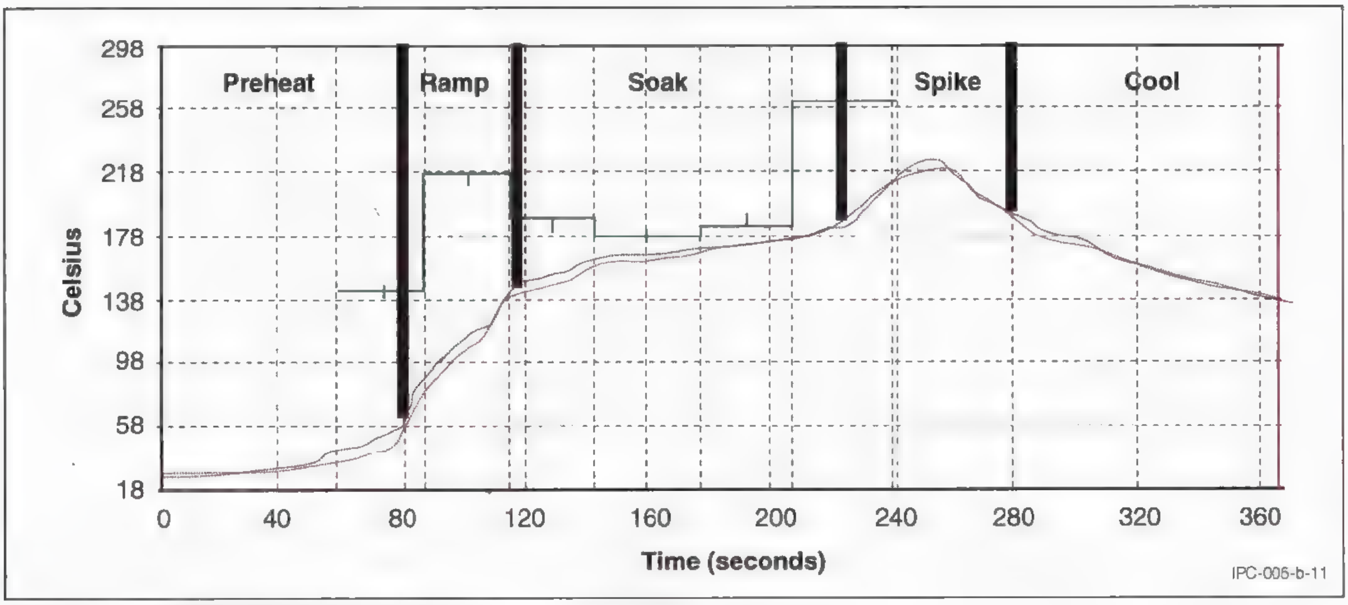

Figure 11 Ramp-Soak-Spike Profile (r-s-s)

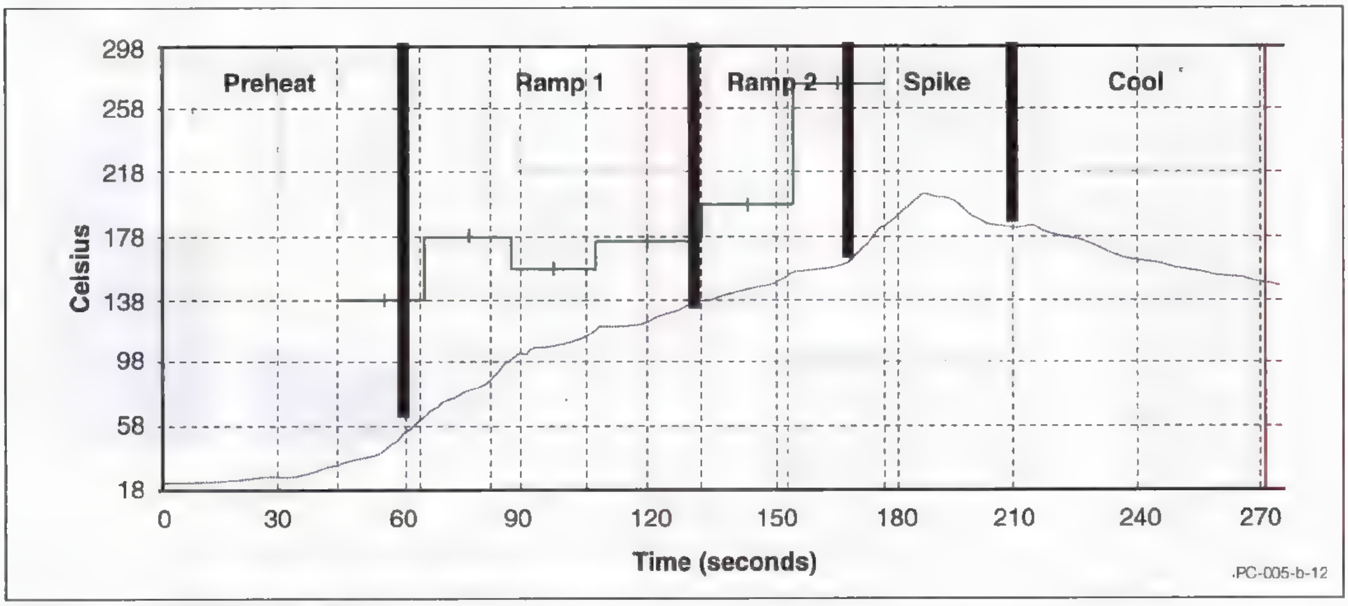

Figure 12 Ramp-Spike Profile (r-s)

Table 7 Ramp Rates for the Two Profiles Used

Profile Type	Ramp-Rate
Ramp-Spike	1.2°C/sec
Ramp-Soak-Spike	1.8 to 2°C/sec

4. Increase in paste viscosity decreases slumping, thereby reducing solder bead formation, which is the case for the ramp-spike profile.

17 Effect of Board Support on Solder Beading

1. As stated earlier the presence/absence of a board support during component placement does not have a significant effect on solder beading. Also, it is a general practice in the industry to use board supports during placement.

2. However the absence of a board support does exhibit a noticeable reduction in beading (Figure 6). The presence of a board support prevents the board from flexing due to the placement pressure.

3. This aspect brings in another variable, namely placement pressure whose effect can be studied in combination with the presence/absence of a board support, by including placement pressure as one of the design variables in a future study.

18 Effect of Component size on Tombstoning

1. Tombstoning is observed only in 0402 components and not observed in 0805 devices. Tombstoning is a phenomenon, caused due to the unequal wetting forces

generated during reflow soldering, between the pads of the same component.

2. It is also understandable that component size geometry, and weight are critical factors in determining the sensitivity of a device to become a tombstone after reflow. This indicates that tombstoning would be increasingly observed with 0201 chip components and hence need to be considered for further studies.

19 Effect of Reflow Profile on Tombstoning

1. For tombstoning the two reflow profiles considered did not exhibit a statistically significant difference in %defects as inferred from the pareto chart (Figure 9).
2. Qualitatively, the ramp-soak-spike profile exhibits lesser tombstones than the ramp-spike profile (Figure 8), which is attributed to the long soak region in the r-s-s profile.
3. A longer duration of soak helps in maintaining a better thermal equilibrium between the component pads, thereby reducing the differential surface tension forces that primarily cause tombstoning.

20 Effect of Stencil Thickness on Tombstoning

1. Stencil thickness did not have a statistically significant influence on tombstoning as inferred from the pareto chart (Figure 9).
2. Studies in the past have shown that having a reduced paste volume (which means a thinner stencil) helps in reducing tombstone occurrence[5]. This fact is evident from Figure 8 where the 4-mil stencil has less % of tombstones than the mil stencil.

21 Effect of Board Support on Tombstoning

1. Board support did not produce a statistically significant difference in the occurrence of tombstones, which is also inferred from Figure 9.
2. From a qualitative standpoint the presence of a board support causes less tombstones than when the support is absent (Figure 8). This behavior is inversed when beading is considered.
3. Depending on the component layout and concentration on a board, appropriate board supports may be provided, that will aid in reducing both solder beading and tombstoning. The positioning should be decided based on the concentration of miniature component on the board.
4. Further studies can be carried out by including board support position as a study variable.

Table 8 summarizes the various factor settings that would minimize solder beading and tombstoning based on the inferences from the analyses:

Table 8 Preferred Settings of Process Parameters

Factors	Preferred factor settings to reduce	
	Solder Beading	Tombstoning
Stencil thickness	4-mils	4-mils
Paste type	Paste-1	Paste-1
Reflow profile	Ramp-Spike (r-s)	Ramp-Soak-Spike (r-s-s)
Board support	Does not cause a significant change	Yes

22 Conclusion

The resolution-V analysis conducted on some of the process variables has identified the factor settings that minimize solder beading and tombstoning. The next step would be to use these as the process settings and perform confirmation runs. Future studies involving, smaller components like 0201's, and parameters such as placement pressure, placement offset, varying board support configurations and their respective interactions, will be conducted to gain a better understanding of the process.

23 References

1. Qi, Q. and Mahajan, R. L., "Analysis of tombstoning phenomenon during reflow," ASME Journal of Electronic Packaging, 1998, vol. 120, no. 1, pp. 61-67.
2. R.K. Sharma, et al., "Process and Design Parameters to Minimize Tombstoning During Surface Mount Reflow," Surface Mount International, 1997.
3. Christina Piasky, Karen Hirasawa, "Design and Assembly Factors for 0201 Components," Pan Pacific Symposium 2001.
4. Edward Kamen, Dan Creveling, Alex Goldstein, and Erin Sahinci, "Analysis of Factors Affecting Component Placement Accuracy in SMT Electronics Assembly," Twenty-Third IEEE/CPMT International Electronics Manufacturing Technology Symposium, 1998, Austin, Texas.
5. Ning-Cheng Lee, "Optimizing Reflow Profile Via Defect Mechanisms Analysis," Surface Mount International, 1998.
6. Koki Solder Paste Manual - Part 3, Koki Company Limited.

APPENDIX C

Presented at IPC SMEMA Council APEX® 2003

A Materials Based Solution for the Elimination of Tombstones

Brian J. Toleno, Ph.D. and Neil Poole, Ph.D.
Henkel Loctite Corporation
Industry, CA

Abstract

The drive for electronic devices to become lightweight, more portable, and posses increased functionality has driven electronic components to smaller and smaller sizes. This decease in size does not only apply to active devices such as chip scale packages (CSPs) and area array devices, but also to discrete components, such as capacitors and resistors. This has lead to an increase in the use of 0402 and 0201 sized components. These components represent a significant decrease in size from other discrete components. The major challenge with these components is typically issues with processing. One of the most common defects associated with these small components is tombstoning. A variety of solutions have been proposed for reducing or eliminating tombstoning. To date these have mostly been changes in design and/or processing. Implementing these changes after production has begun can be costly and time consuming. Also, even if some of these changes are implemented other process factors can contribute to an increase in tombstoning. This paper presents a materials based solution to the tombstoning issue. This solution widens the process window with respect to tombstoning and has been used to reduce the occurrence of tombstones in production. Causes of tombstoning, case studies, and the mechanism behind how the material reduces tombstoning will be presented.

Introduction

Tombstoning (also known as drawbriding, stonehenging, and the Manhattan effect) is used to describe a number of defects, the most common occurs when one termination of a component lifts into the air during reflow (Figure 1A). The effect occurs due to a force imbalance across the component during reflow. Typically, the solder on one side of the component becomes molten before the other side. The high surface tension of the solder combined with the low mass of the component leads to the termination edge opposite that of the molten solder lifting into the air. These unbalanced forces can also cause a component to skew to the side (Figure 1B) or possibly just lift high enough to ride on top of the solder, reducing reliability, but not necessarily causing an open connection (Figure 2). This unbalanced reflow can have several causes. For example, a large trace underneath or adjacent to one side of a component can lead to heat being thieved from one side, causing the other side to reflow first. In this case the open side will be the side adjacent to the thermal thief. This effect can also be observed if there is a large device or a component with a large heat sink near one side of small discrete component (Figure 3). Another way a thermal imbalance can occur across a small component is by the direction of reflow. In cases where there is a sharp thermal gradient across the zones of an oven and a component is reflowed parallel to the direction of the belt, the front edge may reflow before the back edge resulting in a tombstone with the open edge on the backside of the component.

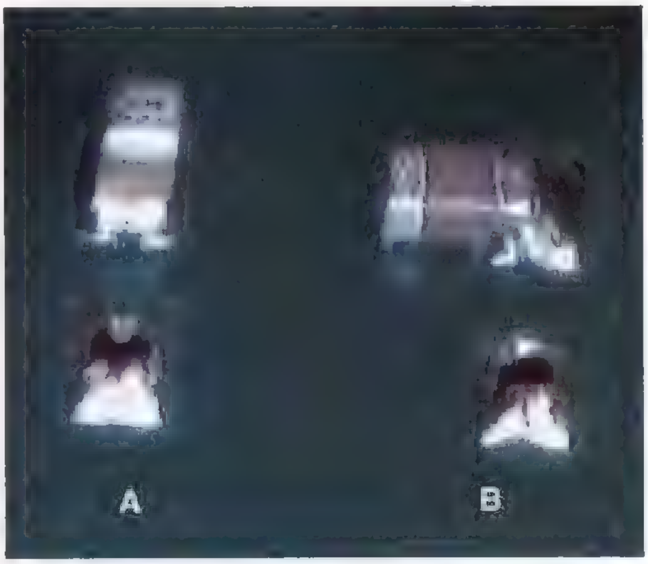

Figure 1 Examples of Tombstoning Defects

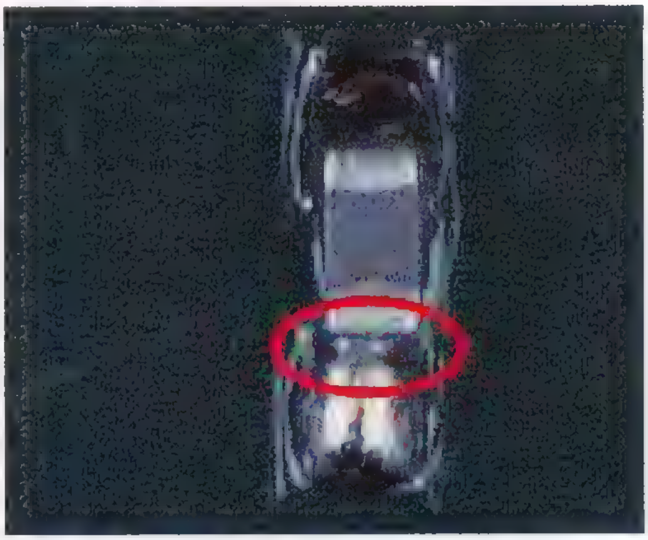

Figure 2 Component Tombstoned without Lifting

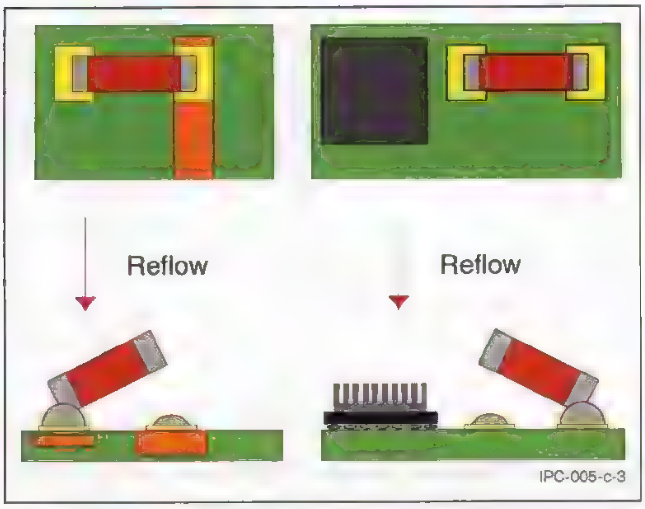

Figure 3 Cartoon of Possible Causes for Tombstoning Defects

Figure 1A illustrates the "classic" tombstone defect. Figure 1B illustrates the less common skewed defect.

In addition to thermal effects, unbalanced placement force may cause tombstoning. If a small component is placed with more force on one side than the other the termination placed deeper into the solder brick stays in contact with the board, and the other termination lifts into the air. These defects are the easiest to observe and identify, but the tombstoning phenomena doesn't always produce these gross defects alone. In some cases the component may be lifted up only enough to have it "ride" on the top of the reflowed solder. In this case the defect may not appear under a quick visual inspection, and if there is physical contact between the component termination and the solder it may even appear to be a valid connection electrically. Although these "defects" pass initial visual and electrical inspection, the long-term reliability of these solder joints are severely compromised. Therefore, these types of defects are sometimes not identified until devices fail in the field. Another related defect is where the component is pulled to the side rather than up into the air (as in Figure 1b). Once again, if the component is up against the side of the reflowed solder the defect is difficult to identify visually and defects may not appear until failures occur in the field.

In order to determine the best methods to reduce the frequency of tombstoning a number designed experiments (DOEs) have been performed that examine process and design effects. Adriance and Schake reported on a design of experiment (DOE) that was used to explore the affects of pad geometry, pad orientation, flux chemistry and processing atmosphere on the tombstoning effect of 0201 components.[1] They concluded that:

- Optimum pad dimensions:
  Width: 0.015″ or 0.018″
- Length: 0.012″
- Spacing between pads: 0.009″
- Components reflowed perpendicular to the application of heat experienced a greater degree of tombstoning.
- Water-soluble paste in air and low-solids paste in nitrogen produced more tombstoning defects than low-solids paste in air.

A DOE was performed[2] to determine the effects of various processes and designs on tombstoning of 0402 and 0201 components.[3] They concluded that combinations of the following parameters reduce tombstoning:

- Low preheat slope and type IV solder paste
- Good solderability finish and low preheat slope
- Good solderability finish and type IV solder paste

Although both papers discuss and present process and design changes in order relieve tombstoning, the second reference does discuss the impact of the solder paste alloy composition and its affect on the propensity to tombstone. Another option that recently became available is utilizing a solder paste designed to reduce the occurrence of tombstoning defects. Ideally, this material would be a drop in replacement for the current solder paste and eliminate tombstoning of components. Use of a material that minimizes tombstoning helps to increase the process window available for production.

In order to reduce the occurrence of tombstoning the concept is to design a material that has a significant melting point (also known as "pasty") range, rather than a sharp melting temperature like eutectic materials (e.g., 63/37 tin/lead). Solder pastes made from non-eutectic alloy powders most commonly include the elements tin, lead and bismuth because this combination has a sufficiently large melting range to produce the desired effect. Typically, they contain only a few percent bismuth and the volume of liquid phase remains low until close to the liquidus temperature of the bulk tin/lead alloy. Other pastes have used a mixture of two alloy powders, for example tin/lead and tin/bismuth, to achieve the same effect. In this case, the lower melting temperature alloy melts first and starts to wet the component, board, and remaining powder. Mutual dissolution between the powders takes place until they form a homogeneous composition before solidification takes place at the end of the reflow process. With both of the bismuth containing solders reliability may be compromised under temperature cycling conditions. At the higher temperatures in the cycle, a low melting eutectic may appear and severely weaken the joint because it is molten. This is just trading one defect (tombstoning) for lower reliability, and is not desirable.

The challenge has been to create the same effective pasty range without compromising the reliability of the joints. Using a mixture of eutectic alloy powders with nearly the same melting temperature achieves this objective if a lower melting temperature ternary composition is not formed on

melting. This is the case if the tin/lead and tin/lead/silver eutectic alloys are used with their melting points of 183° and 179°C respectively. When they coalesce together, the melting range of the final alloy is only 179 - 183°C. This range allows for a wider process window with respect to tombstoning, yet does not comprise reliability in thermal cycling. Attempts to keep the volume of the liquid metal to a minimum before complete reflow by using a mixture that contains more of the high melting temperature powder than the lower melting alloy powder are only partially successful. As the lower melting temperature alloy melts it diffuses into the higher melting particles and causes too large of a liquid volume, reducing the melting temperature of the entire system. This effect may be reduced by using smaller particles of the low melting alloy and thereby restricting the area of contact with the higher melting larger particles,[4] the progress of reflow is illustrated in Figure 4.

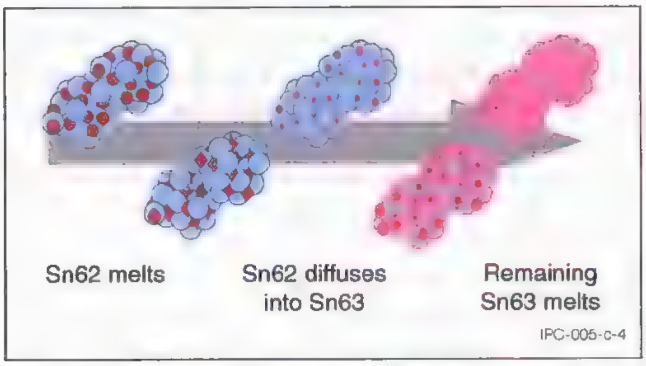

Figure 4 Schematic of the Phased Reflow of Mixed Sn62(●) and Sn63(●) Alloy Powders in Solder Paste

Experimental evidence that this is actually takes place during reflow is shown in the differential scanning calorimitry (DSC) curve Figure 5 which is a plot of heat flow in solder paste as it passes through the melting stage in a typical solder paste reflow profile. The heat flow corresponds to the latent heat of melting of the solder powder (y-axis). The x-axis shows the temperature of the material but this is also a time axis as the solder paste passes through the reflow oven. The eutectic solder paste (Sn62) takes a finite time to melt because the system is changing too fast for equilibrium to be established. Nevertheless, it changes from totally solid to totally liquid in a shorter time than the mixed solder powder anti-tombstone blend (63S4). This extra time when the alloy is in a pasty condition is sufficient to prevent tombstone defects when they would otherwise occur with the eutectic solder paste.

Experimental

The affect of the solder powder formulation change was tested by comparing 0402 components assembled under identical conditions, including identical flux medium. For this experiment a PCB, Figure 6, was used that had a variety of 0402 pad geometries and orientations, Figure 7. The factors varied include:

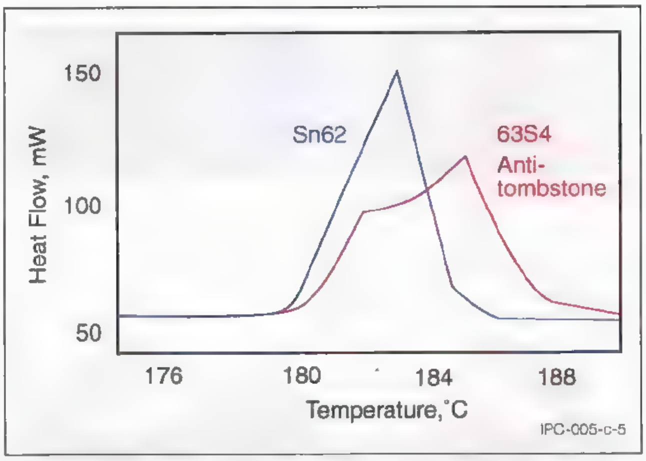

Figure 5 Differential Scanning Calorimetry for Two Solder Pastes Passing through the Melting Stage in a Typical Reflow Process Temperature Profile

1. Pad geometry
   a. Square (.020 X .020, .030 X .030)
   b. Rectangular (.020 X .030, .030 X .020, .040 X .030, .040 X .020)
2. Space Between Pads
   a. .023
   b. .016
   c. .011
3. Direction of Reflow
   a. Vertical
   b. Horizontal

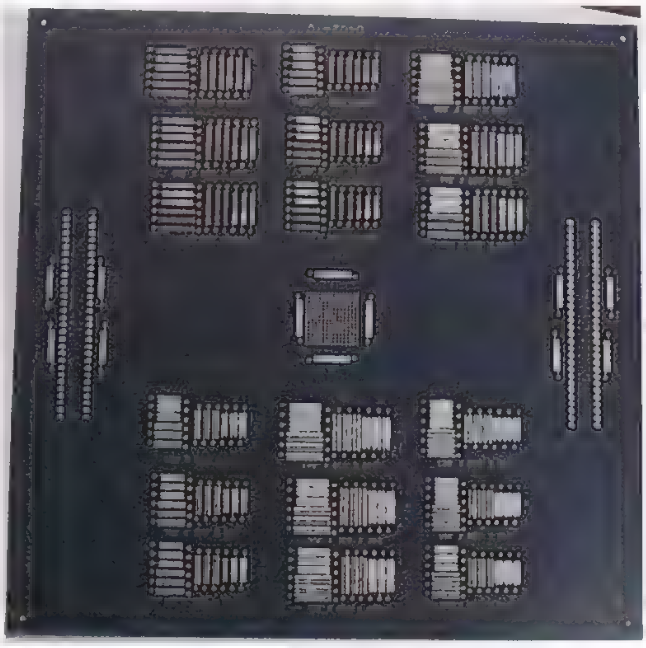

Figure 6 Photograph of Test Vehicle Used

The various combinations examined are listed in Table 1.

For each combination of variables, on a single PCB, there were 18 sites; two boards were run, producing 36 defect opportunities for each combination of variables, and a total

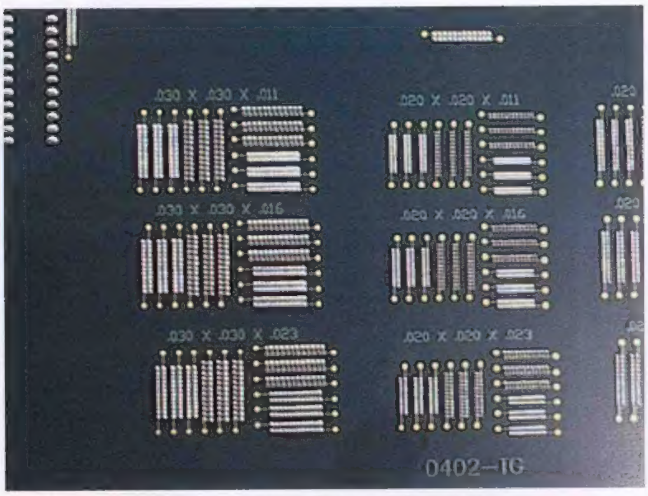

Figure 7 Photograph of Pads Examined for Anti-Tombstoning Trial

of 2592 opportunities per material. Boards were assembled at the Engent (formally the Siemens Production Laboratory) facility using the following equipment: Siemens Siplace HS-50 and F-5, Siemens Siplace SP-500 Screen Printer, and Siemens Siplace RN-38 Reflow Oven. Prior to component placement solder prints were verified using an ASC International VisionMaster AP212 Measurement System. Two different solder pastes were used in this trial. In both cases a no-clean, high-speed, Sn/Pb eutectic solder paste was used (Flux NC1). The only difference was the composition of the alloy. In the case of the control solder paste the solder powder alloy is a Type III power size. The anti-tombstoning solder paste contains a mixture of alloy as described in the introduction (Sn63/Pb37 and Sn62/Pb36/Ag2) and powder sizes. After reflow the boards were examined under a microscope and the number of defects counted.

In addition to the laboratory experiments, the effectiveness of this material to reduce the number of defects was observed on an actual production line. The manufacturing line consisted of a DEK 265 screen printer, Siemens Pick and Place machine, and a ten zone Omni Flow reflow oven. In both cases the same flux chemistry (NC2) and reflow profile was used.

Results

Without looking at the specific location of the defects, the anti-tombstoning solder paste formulation reduced the number of defects by half, Figure 8. A comparison of the number of defects with respect to spacing (Figure 9), orientation (Figure 10), and geometry (Figure 11) show irregardless of any other effects, the anti-tombstoning solder paste reduces the number of defects. In order to validate this observation an analysis of variance was performed on the data collected using the MINITAB[5] software package, the results are presented in Table 2.

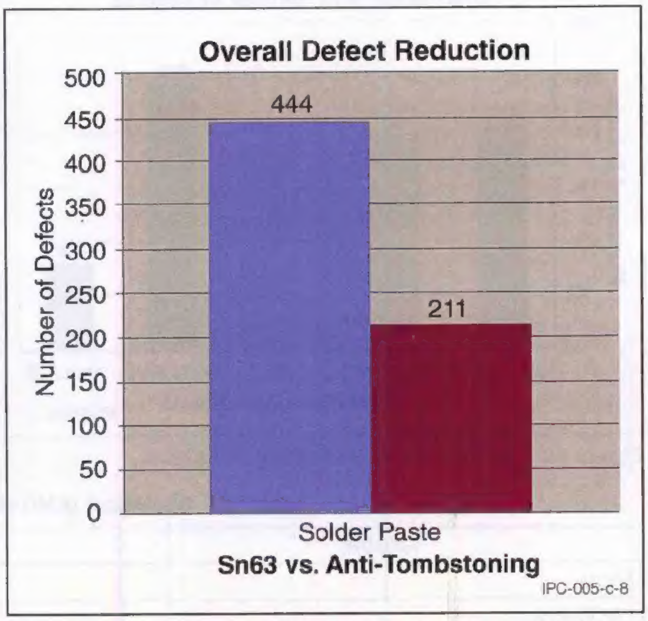

Figure 8 Overall Defect Reduction using New Material

Table 1 Pad Dimensions (L x W x spacing) and Orientation (H or V) Examined

.030 X .030 X .023 (V)	.020 X .030 X .023 (H)	.040 X .020 X .023 (H)
.030 X .030 X .023 (H)	.020 X .030 X .023 (V)	.040 X .020 X .023 (V)
.030 X .030 X .016 (V)	.020 X .030 X .016 (V)	.040 X .020 X .016 (V)
.030 X .030 X .016 (H)	.020 X .030 X .016 (H)	.040 X .020 X .016 (H)
.030 X .030 X .011 (H)	.020 X .030 X .011 (H)	.040 X .020 X .011 (H)
.030 X .030 X .011 (V)	.020 X .030 X .011 (V)	.040 X .020 X .011 (V)
.020 X .020 X .023 (H)	.030 X .020 X .023 (H)	.040 X .030 X .023 (H)
.020 X .020 X .023 (V)	.030 X .020 X .023 (V)	.040 X .030 X .023 (V)
.020 X .020 X .016 (V)	.030 X .020 X .016 (V)	.040 X .030 X .016 (V)
.020 X .020 X .016 (H)	.030 X .020 X .016 (H)	.040 X .030 X .016 (H)
.020 X .020 X .011 (H)	.030 X .020 X .011 (H)	.040 X .030 X .011 (H)
.020 X .020 X .011 (V)	.030 X .020 X .011 (V)	.040 X .030 X .011 (V)

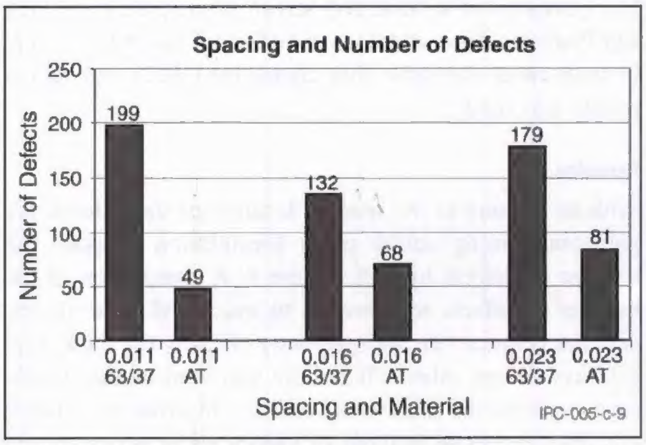

Figure 9 Spacing Effect on Number of Defects

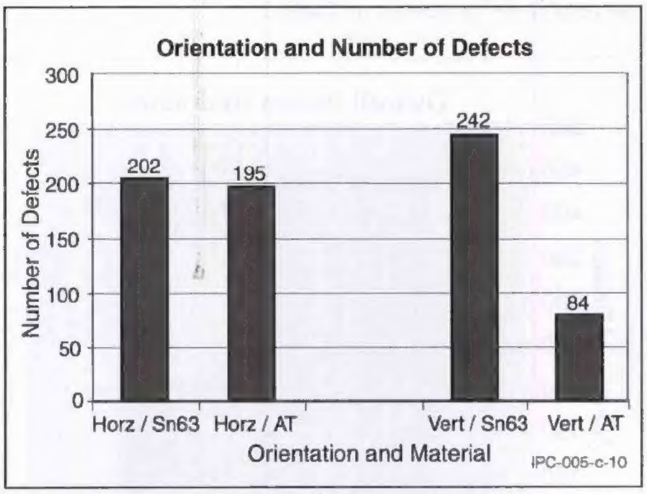

Figure 10 Orientation and Number of Defects

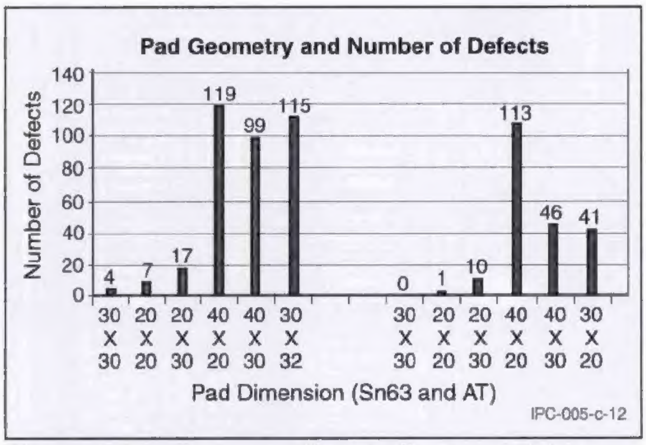

Figure 11 Pad Geometry Effect on Number of Defects

The terms in Table 2 are the variable and the variable interactions, degrees of freedom (DF), sum of squares (SS), and the P value (P). The interactions are identified by variable separated with a *. The highlighted rows indicate the significant terms in the model used (rows where the P values are less than 0.050). Notice that although spacing and orientation are not significant on their own, their interaction is significant. The two most significant factors in reducing the number of defects in this experiment are pad size and the solder paste used.

Beyond the controlled conditions of the laboratory environment the anti-tombstoning alloy blend (with a different no-clean formulation) was used to reduce defects in a production environment. This manufacturer was experiencing

Table 2 Statistical (ANOVA) Analysis of Defects Observed

Variable	DF	SS	P
Paste	1	377.01	0.003
Geometry	5	2425.37	0.000
Spacing	2	54.51	0.500
Orientation	1	0.06	0.968
Paste*Geometry	5	584.20	0.016
Paste*Spacing	2	13.43	0.842
Paste*Orientation	1	47.84	0.272
Geometry*Spacing	10	671.15	0.092
Geometry*Orientation	5	425.48	0.065
Spacing*Orientation	2	381.29	0.010
Paste*Geometry*Spacing	10	38.24	1.000
Paste*Geometry*Orientation	5	64.53	0.893
Paste*Spacing*Orientation	2	14.26	0.833
Geometry*Spacing*Orientation	10	692.04	0.081
Paste*Geometry*Spacing*Orientation	10	90.74	0.922
Error	72	2807.50	
Total	143	8687.66	

a rate of 10 tombstones on an assembly with 95 0402 components. They were reworking the components by hand after exiting the reflow oven. The anti-tombstoning solder paste was then substituted for the solder paste currently being used on the line (Flux NC2). ***Without changing any parameters*** the number of tombstones per board was reduced to zero. A run of twenty assemblies, produced ***zero*** defects (a decrease from 200 defects on average).

Conclusions

In the manufacturing study performed the second most significant factor in tombstoning was the solder paste used. This shows that the use of an anti-tombstoning alloy can significantly reduce the rate of tombstoning defects. Furthermore, the ability of this alloy blend to reduce tombstoning was shown to work with two different no-clean flux chemistries.

In order to reduce and eliminate tombstoning defects from final assemblies steps can be taken during design and processing. In cases where these defects still occur a drop in solder paste replacement designed to reduce tombstones will further reduce the rate of defects.

Acknowlegements

The authors would like to thank Paul Houston, Ealy Brandon, Dan Baldwin, and the members of Engent, and Doug Dixon and Brian Betti of Loctite.

References

1. J. H. Adriance and J. D. Schake, Universal Instruments Corp., APEX 2000 Proceedings, P-EQ4/3-1, April 2000.
2. Michael Yuen et al. from Plexus and Andy Mackie from Praxair
3. M. Yuen, H. Benedict, K. Havlovitz, and T. Pitsch, Plexus Corporation and A. C. Mackie, Praxair Inc., www.plexus.com.
4. M. Warwick, "Tombstoning Reduction Via Phased-Reflow Soldering." SMT, September 2002.
5. MINITAB Version 12, Minitab, Inc., State College, PA

IPC
ASSOCIATION CONNECTING ELECTRONICS INDUSTRIES ®

ANSI/IPC-T-50 Terms and Definitions for Interconnecting and Packaging Electronic Circuits Definition Submission/Approval Sheet

The purpose of this form is to keep current with terms routinely used in the industry and their definitions. Individuals or companies are invited to comment. Please complete this form and return to:

IPC
3000 Lakeside Drive, Suite 309S
Bannockburn, IL 60015-1219
Fax: 847 615.7105

SUBMITTOR INFORMATION:

Name: ______________________

Company: ______________________

City: ______________________

State/Zip: ______________________

Telephone: ______________________

Date: ______________________

❑ This is a **NEW** term and definition being submitted.
❑ This is an **ADDITION** to an existing term and definition(s).
❑ This is a **CHANGE** to an existing definition.

Term	Definition

If space not adequate, use reverse side or attach additional sheet(s).

Artwork: ❑ Not Applicable ❑ Required ❑ To be supplied
❑ Included: Electronic File Name: ______________________

Document(s) to which this term applies: ______________________

Committees affected by this term: ______________________

Office Use

IPC Office

Date Received: ______________________
Comments Collated: ______________________
Returned for Action: ______________________
Revision Inclusion: ______________________

Committee 2-30

Date of Initial Review: ______________________
Comment Resolution: ______________________
Committee Action: ❑ Accepted ❑ Rejected
❑ Accept Modify

IEC Classification

Classification Code • Serial Number

Terms and Definition Committee Final Approval Authorization:

Committee 2-30 has approved the above term for release in the next revision.

Name: ______________________ Committee: IPC 2-30 Date: ______________________